The mixing characteristics of dilution jets issuing into a confined cross-flow

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

Additional Information:


Metadata Record: https://dspace.lboro.ac.uk/2134/32627

Publisher: © J.F. Carrotte

Rights: This work is made available according to the conditions of the Creative Commons Attribution-NonCommercial-NoDerivatives 4.0 International (CC BY-NC-ND 4.0) licence. Full details of this licence are available at: https://creativecommons.org/licenses/by-nc-nd/4.0/

Please cite the published version.
<table>
<thead>
<tr>
<th>AUTHOR/FILING TITLE</th>
<th>CARRUTHE, J F</th>
</tr>
</thead>
<tbody>
<tr>
<td>ACCESSION/COPY NO.</td>
<td>036001995</td>
</tr>
<tr>
<td>VOL. NO.</td>
<td>CLASS MARK</td>
</tr>
<tr>
<td>13 DEC 1935</td>
<td>Loan Copy</td>
</tr>
<tr>
<td>1-5 DEC 1995</td>
<td></td>
</tr>
<tr>
<td>8 MAR 1996</td>
<td></td>
</tr>
<tr>
<td>26 APR 1996</td>
<td></td>
</tr>
</tbody>
</table>
THE MIXING CHARACTERISTICS OF DILUTION JETS ISSUING

INTO A CONFINED CROSS-FLOW

by

J.F. Carrotte

Submitted for the degree of Doctor of Philosophy in the Department of Transport Technology, Loughborough University

May 1990
ERRATUM

<table>
<thead>
<tr>
<th>Page</th>
<th>Line</th>
<th>From : To</th>
</tr>
</thead>
<tbody>
<tr>
<td>ii)</td>
<td>28</td>
<td>holes were modified : holes was modified</td>
</tr>
<tr>
<td>2</td>
<td>34</td>
<td>of 1300K : below 1300K</td>
</tr>
<tr>
<td>3</td>
<td>28</td>
<td>jets are injected : jets is injected</td>
</tr>
<tr>
<td>4</td>
<td>16</td>
<td>falls below : penetrates less than</td>
</tr>
<tr>
<td>13</td>
<td>32</td>
<td>dilution holes could : dilutions holes that</td>
</tr>
<tr>
<td>13</td>
<td>33</td>
<td>annulus be minimised : annulus could be minimised</td>
</tr>
<tr>
<td>14</td>
<td>25</td>
<td>as was also : as also</td>
</tr>
<tr>
<td>15</td>
<td>9</td>
<td>is to initially : was initially</td>
</tr>
<tr>
<td>17</td>
<td>1</td>
<td>heading : The test facility</td>
</tr>
<tr>
<td>18</td>
<td>3</td>
<td>this also eliminates : this eliminates</td>
</tr>
<tr>
<td>21</td>
<td>8</td>
<td>can only can only : can only</td>
</tr>
<tr>
<td>27</td>
<td>19</td>
<td>$h = C_p T : \frac{dh}{dT} = C_p dT$</td>
</tr>
<tr>
<td>31</td>
<td>9</td>
<td>more preferable : preferable</td>
</tr>
<tr>
<td>36</td>
<td>11</td>
<td>a limited amount of measurements : limited measurements</td>
</tr>
<tr>
<td>41</td>
<td>18</td>
<td>program : programme</td>
</tr>
<tr>
<td>48</td>
<td>2</td>
<td>possible though : possible</td>
</tr>
<tr>
<td>50</td>
<td>3</td>
<td>program : programme</td>
</tr>
<tr>
<td>50</td>
<td>3</td>
<td>to what extent these : the extent to which</td>
</tr>
<tr>
<td>58</td>
<td>32</td>
<td>what : the</td>
</tr>
<tr>
<td>60</td>
<td>10</td>
<td>means : is such</td>
</tr>
<tr>
<td>64</td>
<td>9</td>
<td>was indicated : indicated</td>
</tr>
</tbody>
</table>
Dedication

To family, friends and colleagues, and the World's greatest sport.
An experimental investigation has been carried out into the mixing of a row of jets injected into a confined cross-flow. Measurements were made on a fully annular test facility, the geometry of the rig simulating that found in the dilution zone of a gas turbine combustion chamber. A small temperature difference of $44^\circ$C between the cross-flow and dilution fluid allowed the mixing characteristics to be assessed, with hot jets being injected into a relatively cold cross-flow at a jet to cross-flow momentum flux ratio of 4.0. The investigation concentrated on differences in the mixing of individual dilution jets, as indicated by the regularity of the temperature patterns around the cross-flow annulus. Despite the uniform conditions approaching the dilution holes there were significant differences in the temperature patterns produced by the dilution jets around the annulus.

When air is supplied to the jets from a feed annulus, measurements with miniature 5 hole pressure probes of the flow issuing from the dilution holes indicated significant variations in the velocity profiles from one hole to another. Vortices and other complex flow fields issue through the holes and both mean velocity and temperature measurements in a developing jet have shown how such features affect the temperature distribution. In particular, the trajectory of fluid within each jet is altered along with the vortex flow field which develops downstream of the hole and this leads to a distortion of the associated temperature distribution. Since the velocity profile varies from one hole to another, so each jet has its own mixing characteristics thereby producing overall irregularity of the temperature pattern around the cross-flow annulus. Furthermore, the detailed flow measurements revealed significant variations between the structure of a single jet, issuing into a relatively unconfined cross-flow, and that of a multiple jet configuration in a confined cross-flow. This is due to several effects including the influence of adjacent holes on the cross-flow surrounding a jet, the confining nature of the cross-flow annulus and the method of supplying air the row of holes.

With similar approach and operating conditions the geometry of the plunged dilution holes were modified in an attempt to make the subsequent development of each jet less sensitive to the effects of supplying fluid from a feed annulus. Tests showed a more uniform mixing of the jets around the modified sector, as indicated by an improvement in the overall regularity of the temperature pattern. At the operating conditions tested, these modified hole designs also exhibited virtually the same discharge coefficient values as that of the standard plunged holes.
Acknowledgements

This work was carried out in the Airflow laboratory at the Department of Transport Technology, Loughborough University and was financially supported by the Royal Aerospace Establishment, Pyestock, contract nos. D/ER1/9/4/2170/113/RAE(P). The author would also like to express his appreciation to Messrs. G. Hodson, R. Marson and D. Glover for their technical assistance in the building of the test rig and instrumentation hardware.

The research was supervised by Mr. A.P. Wray whose guidance and contribution to the work is much appreciated. In particular, his 5 hole probe analysis techniques that have been used for velocity measurements together with his advice on the mechanical rig design is gratefully acknowledged.

As overall director of research, a special acknowledgement is given to Professor S.J. Stevens who has always been approachable, enthusiastic and provided valuable advice throughout. His encouragement to publish results and overall interest was invaluable and a major factor in the completion of this work.

Help and advice has also been given by other members of the research staff at the Department of Transport Technology, including my office colleagues M.A. Passmore and K.F. Young.

It is to all the aforementioned personnel that their help and attitude, along with their good sense of humour, is gratefully acknowledged and with whom it has been a pleasure to work.

And finally to all my family, friends and colleagues who have had to endure my company during the writing of this thesis - thank you for your patience and understanding!
List of Contents

<table>
<thead>
<tr>
<th>Section</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>Dedication</td>
<td>(i)</td>
</tr>
<tr>
<td>Summary</td>
<td>(ii)</td>
</tr>
<tr>
<td>Acknowledgements</td>
<td>(iii)</td>
</tr>
<tr>
<td>List of Contents</td>
<td>(iv)</td>
</tr>
<tr>
<td>List of Figures</td>
<td>(vi)</td>
</tr>
<tr>
<td>Nomenclature</td>
<td>(x)</td>
</tr>
</tbody>
</table>

Chapter 1  Introduction

1.1 Mixing of dilution jets in gas turbine combustors 2
1.2 General review of jet mixing
   1.2.1 Single jet in a cross-flow 4
   1.2.2 General flow characteristics of a single jet 5
   1.2.3 Prediction methods 7
   1.2.4 Multiple jets in a confined cross-flow 10
1.3 Review of work specific to this investigation 13
1.4 Objectives and scope of the present investigation 15

Chapter 2  Test Facility

2.1 Description of the test facility 17
2.2 Instrumentation 19
2.3 Rig traversing
   2.3.1 Complete annular surveys 22
   2.3.2 Sector surveys 23
2.4 Rig operating conditions 24

Chapter 3  Data Analysis and Performance Parameters

3.1 Data reduction 27
3.2 Performance parameters 31
3.3 Vorticity definitions 36
3.4 Presentation of data 37
3.5 Estimation of experimental errors
   3.5.1 Temperature measurements 39
   3.5.2 Velocity measurements 41
**List of Figures**

<table>
<thead>
<tr>
<th>Fig.</th>
<th>Title</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.1</td>
<td>Types of combustion chambers</td>
<td>81</td>
</tr>
<tr>
<td>1.2</td>
<td>Main features of a combustion chamber</td>
<td>82</td>
</tr>
<tr>
<td>1.3</td>
<td>Jet velocity and temperature centrelines - Kamotani and Greber[5]</td>
<td>83</td>
</tr>
<tr>
<td>1.4</td>
<td>Influence of injection angle on jet trajectory - Platten and Keffer[1]</td>
<td>84</td>
</tr>
<tr>
<td>1.5</td>
<td>Development of a jet - Abramovich[3]</td>
<td>85</td>
</tr>
<tr>
<td>1.6</td>
<td>Development of a jet - Andreopoulos and Rodi[14]</td>
<td>86</td>
</tr>
<tr>
<td>1.7</td>
<td>Integral prediction method for a single jet - Abramovich[3]</td>
<td>87</td>
</tr>
<tr>
<td>1.8</td>
<td>Integral prediction method for a single jet - Adler and Baron[19]</td>
<td>88</td>
</tr>
<tr>
<td>1.9</td>
<td>Measured and predicted velocity distributions - Adler and Baron[19]</td>
<td>89</td>
</tr>
<tr>
<td>1.10</td>
<td>Measured and predicted streamwise (U) velocity profiles - Sykes et. al.[17]</td>
<td>90</td>
</tr>
<tr>
<td>1.11</td>
<td>Predicted temperature centrelines of single and multiple jets</td>
<td>91</td>
</tr>
<tr>
<td>1.12</td>
<td>Temperature distribution produced by 45° angled slots - Srinivasan et. al.[22]</td>
<td>92</td>
</tr>
<tr>
<td>1.13</td>
<td>Measured and predicted temperature distributions - Srinivasan and White[23]</td>
<td>93</td>
</tr>
<tr>
<td>1.14</td>
<td>Typical sector temperature pattern at exit of an annular combustor (T/T_{mean})</td>
<td>94</td>
</tr>
</tbody>
</table>

2.1 The test facility
   a) Working section                                           | 95   |
   b) Airflow paths                                             | 96   |
   c) Injection plane geometry                                  | 97   |
2.2 Dilution hole manufacture                                   | 98   |
2.3 The test facility - Bailey's[41] and Wilson's[42] configuration | 99   |
2.4 Thermocouple geometry                                      | 100  |
2.5 Five hole probe geometry                                    | 101  |
2.6 Pitot probe rakes                                          | 102  |
2.7 Data acquisition system - complete annular temperature survey | 103  |
2.8 Method of thermocouple positioning - complete annular surveys | 104  |
2.9 Data acquisition system
   a) Sector temperature survey                                 | 105  |
   b) Sector 5 hole probe survey                                 | 106  |
List of Figures

3.1 Method of temperature correction 107
3.2 Method of pressure correction 108
3.3 Block temperature distribution 109
3.4 Jet distortion parameter 110
3.5 Presentation of cylindrical data grid 111

4.1.1 Cross-flow annulus axial velocity profiles (X/D=-6.0) 112
4.1.2 Feed annulus axial velocity profiles (X/D=-3.5) 113
4.2.1 Radial variation of temperature factors (X/D=2.0) 114
4.2.2 Temperature distributions downstream of holes 1 to 16 (X/D=2.0) 115
4.2.3 Temperature distributions recorded by Wilson[42] (X/D=2.0) 116
4.2.4 Temperature distributions recorded by Wilson[42] (X/D=3.0) 117
4.2.5 Temperature distributions recorded by Wilson[42] (X/D=4.0) 118
4.2.6 Temperature distributions downstream of holes 7 to 12 (X/D=2.0) 119
4.2.7 Temperature distributions for holes 9 and 10 120
4.3.1 Velocity distributions at the exit planes of holes 7 to 12 (Y/D=0.05)
  - Without cross-flow 121
4.3.2 Velocity distributions at the exit planes of holes 9 and 10 (Y/D=0.05)
  - Without cross-flow 122
4.3.3 Velocity distributions at the exit planes of holes 9 and 10 (Y/D=0.05)
  - With cross-flow 123
4.3.4 Pitch angle variation of the flow issuing through a dilution hole (Y/D=0.05)
  a) Hole 8 124
  b) Hole 10 124
4.4.1 Velocity vectors downstream of holes 7 to 12 (X/D=2.0) 125
4.4.2 Velocity and temperature distributions
  a) Hole 8, X/D=2.0 126
  b) Hole 10, X/D=2.0 127
4.4.3 Flow fields downstream of a jet in a cross-flow
  a) Single jet 128
  b) Multiple jet 128
4.5.1 Velocity and temperature distribution (Hole 8, X/D=0.30) 129
4.5.2 Velocity and temperature distribution (Hole 8, X/D=0.54) 130
4.5.3 Velocity vector plot (Hole 8, Y/D=0.20) 131
4.5.4 Streamwise (U/Vp) velocity component profiles (Hole 8, Y/D=0.20) 132
4.5.5 Streamwise and circumferential velocity component profiles
  (Hole 8, X/D=0.54) 133
4.5.6 Temperature distribution (Hole 8, Y/D=0.20) 134
List of Figures

4.5.7 Velocity and temperature distribution (Hole 8, X/D=2.0) 135
4.5.8 Velocity vector plot (Hole 10, Y/D=0.20) 136
4.5.9 Velocity and temperature distribution (Hole 10, X/D=0.54) 137
4.5.10 Velocity and temperature distribution (Hole 10, X/D=0.72) 138
4.5.11 Velocity and temperature distribution (Hole 10, X/D=2.0) 139
4.5.12 Temperature distribution (Hole 10, Y/D=0.20) 140
4.6.1 Characteristics of multiple jets in a cross-flow 141
4.6.2 Streamwise vorticity distribution ($\Omega_{x/D/V_j}$)
   a) Hole 8, X/D=0.30 142
   b) Hole 8, X/D=2.0 142
4.6.3 Radial vorticity distribution ($\Omega_{y/D/V_j}$)
   (Hole 8, X/D=0.30) 143
4.6.4 Influence of injection wall on the flow field downstream of a jet
   a) Single jet 144
   b) Multiple jet 144
4.6.5 Velocity vector plot (X/D=2.0) 145
4.7.1 Water analogy rig observations of flow feeding a dilution hole 146
4.7.2 Velocity vector plots for holes 4 to 8 (Y/D=0.05)
   - without cross-flow 147
4.7.3 Flow visualisation results of vortex formation 148
4.7.4 Instability of flow feeding a hole 149
4.7.5 Streamline curvature effects 150
4.7.6 Formation of Görtlert vortices on concave surface 151
4.7.7 Velocity vectors for holes 9 and 11 (Y/D=0.05) - without cross-flow
   a) Hole spacing to diameter ratio (S/D) = 2.75 152
   b) Hole spacing to diameter ratio (S/D) = 5.50 152
4.7.8 Temperature distributions downstream of holes 9 and 11 (X/D=2.0)
   a) Hole spacing to diameter ratio (S/D) = 2.75 153
   b) Hole spacing to diameter ratio (S/D) = 5.50 153
4.7.9 Flow control devices 154
4.8.1 Modified dilution holes 155
4.8.2 Temperature distributions - Design A components (X/D=2.0) 156
4.8.3 Jet distortions - Design A components 157
4.8.4 Velocity vector plot for standard hole with back-stop (Y/D = 0.35) 158
4.8.5 Temperature distributions (Y/D=0.35) 159
4.8.6 Temperature distributions - Design B (X/D=2.0) 160
4.8.7 Jet distortion - Design B 161

(viii)
List of Figures

4.8.8  Comparison of standard and Design B downstream flow fields (X/D=2.0)
    a) Velocity vector plots 162
    b) Streamwise vorticity distribution 162

4.8.9  Temperature distributions downstream of holes 7 to 12
    (Design A, X/D=2.0) 163

4.8.10 Temperature distributions downstream of holes 7 to 12
     (Standard holes, X/D=2.0) 164

4.8.11 Temperature distribution factors
    a) Standard holes 165
    b) Design A geometry 165

4.8.12 Temperature asymmetry factor for Design A and standard hole sectors 166

4.8.13 Standard deviations for Design A and standard hole sectors 167

4.8.14 Hole discharge coefficients 168

A.1.1  Definition of circulation 173

A.1.2  Vortexlines and vortextubes 174

A.2.1  Five hole probe geometry and nomenclature 179
Nomenclature

A  area
A_h  geometric area of dilution hole
A_ref  reference hole area (=\pi D^2/4)
b  characteristic width of jet
C_d  hole discharge coefficient (m_h=C_dA_h(2\rho(P-p_a))^{0.5})
C_n  drag coefficient
C_p  specific heat at constant pressure
D  reference diameter of standard holes
D_e  effective hole diameter (=4A_h/\pi)^{0.5}
D_j  jet diameter
F_n  pressure force normal to jet centre-line
F_c  centripetal force normal to jet centre-line
H  mixing duct height
h  enthalpy
I_1  \int u dA
I_2  \int u^2 dA
J  momentum flux ratio (=\rho_j V_j^2/\rho_c V^2) or number of jets
K  circulation
m_h  mass flow through dilution hole
m'  mass flow through local control volume
N  number of data points
P  total pressure in feed annulus
p  pressure
P_\alpha  hole discharge plenum pressure
q  dynamic head (0.5pu^2)
Re  Reynolds number
R_f  thermocouple recovery factor
R_p  radius of plunging on dilution holes
r  radius
r_i  inner radius of annulus
r_n  non-dimensional radius (=r-r_i)/(r_o-r_i)
r_o  outer radius of annulus
S  distance between dilution holes
T  corrected temperature
TAF  temperature asymmetry factor
TDF  temperature distribution factor

(x)
Nomenclature

\( T_0 \) reference cross-flow temperature (11°C)
\( T_j \) reference jet temperature (55°C)
\( t \) casing thickness
\( u \) velocity
\( U,V,W \) velocities in X,Y,Z directions
\( V_c \) velocity of cross-flow
\( V_j \) mean velocity of jet
\( X \) co-ordinate measured from hole center in downstream direction
\( Y \) co-ordinate measured normal to injection wall in radial direction
\( Z \) co-ordinate measured from hole centre in a circumferential direction
\( \alpha \) pitch angle of jet measured from radial (Y) axis
\( \delta^* \) displacement thickness (=\( \int (1-u/U)dy \))
\( \phi^* \) angle measured from the centre of a dilution hole
\( \theta \) angle
\( \theta \) non-dimensional temperature (=\( [T-T_c]/[T_j-T_c] \))
\( \theta \) momentum thickness (=\( \int [u/U][1-u/U]dy \))
\( \theta' \) non-dimensional temperature profile (=\( [T_{max}-T_c]/[T_j-T_c] \))
\( \rho \) density
\( \sigma \) sector standard deviation
\( \sigma_j \) jet distortion
\( \nu \) effective kinematic viscosity
\( \Omega_x, \Omega_y, \Omega_z \) vorticity in X,Y,Z directions

Subscripts

\( b \) block temperature value
\( c \) cross-flow
\( j \) jet
\( \text{max} \) maximum value
\( \text{meas} \) measured value
\( r \) radius

Superscripts

\( - \) mean
Chapter 1: INTRODUCTION

1.1 Mixing of dilution jets in gas turbine combustor 2

1.2 General review of jet mixing 3
   1.2.1 Single jet in a cross-flow 4
   1.2.2 General flow characteristics of a single jet 5
   1.2.3 Prediction methods 7
   1.2.4 Multiple jets in a confined cross-flow 10

1.3 Review of work specific to this investigation 13

1.4 Objectives and scope of the present investigation 15
1.1 Mixing of Dilution Jets in Gas Turbine Combustion Chambers

Since the running of the first experimental prototype in 1937, the gas turbine engine has become widely used in aircraft propulsion due to its ability to produce large amounts of power and low weight relative to other power-plants. The gas turbine engine basically consists of a compressor, which supplies air to a combustion chamber where fuel is injected and combustion takes place, so adding thermal energy to the flow which is then expanded through a turbine.

For aircraft engines where size and weight are important, several different combustor geometries are used (Fig.1.1), the actual design selected being dependent on the application concerned. For gas turbines using centrifugal compressors a number of tubular combustion chambers are often employed whilst an annular chamber is more suitable for use on larger engines with axial flow compressors. An alternative to this is the tubo-annular design which attempts to combine some of the best features of both the aforementioned systems by placing a number of cylindrical liners in a single annular casing. Despite the different geometries of these designs however, their common requirements mean that the same operating features are present in each combustion system.

Since the aerodynamic pressure loss of the combustor is proportional to the square of the airspeed, the flow at exit from the compressor is initially diffused prior to it entering the combustion chamber (Fig.1.2). Some of the air then passes directly through the head of the flame tube (or liner) mounted in the combustion chamber, whilst the remainder of the air passes into the annuli formed by the liner and pressure casings. These annuli supply air to perforations in the liner through which the air passes, thereby creating patterns of jets which form 3 distinct regions within the flame tube, each region fulfilling different objectives. The opposed jets of air formed by the first row of liner holes together with the flow passing through the flame tube head creates a large recirculating flow in the primary zone. The amount of air is controlled so that a suitable equivalence ratio is achieved with the fuel injected into the primary zone, and a low velocity region is thereby created within which combustion can be sustained. Further air is injected through the liner into the intermediate zone to permit completion of the combustion process, whilst also serving to recover dissociation losses which can occur at certain operating conditions.

The temperature of the combustion products on completion of the combustion process is far too great for the materials in the highly stressed turbine stage to operate in. For example, in the case of a large turbofan engine such as the RB211 the turbine blade material must be maintained at a temperature of 1300K, whereas the primary zone exit temperature can be as high as 2500K. Although coating and local cooling of the blade surface allows the
material to operate in gas temperatures of up to 1650K, a significant reduction in the temperature of the flow issuing from the combustor primary zone is still required. The function of the dilution zone is therefore to dilute the hot gases leaving the combustor so that an outlet temperature distribution is produced which is consistent with the integrity of the downstream turbine stage. This requires minimum temperatures at the turbine root where material stresses are highest, and at the turbine tip where the sealing materials need to be protected. Relatively cold jets of air are therefore injected into the combustion products, these being formed by a row of holes in the flame tube liner. The number, size and arrangement of the dilution holes together with the velocity of the gases passing through them are some of the major factors by which the radial and circumferential temperature profiles at the exit of a combustion chamber can be controlled.

The efficiency of the gas turbine cycle is dependent upon the maximum cycle temperature achieved along with the pressure ratio of the engine. Thus, over the last 40 years the demand for improved performance, in terms of higher engine thrust to weight ratio and lower specific fuel consumption, has resulted in a continuing pressure for greater turbine inlet temperatures and reductions in the length of the combustion system. The higher turbine inlet temperatures infer the need for the dilution zone to achieve closer adherence to the design temperature profiles, but the higher temperatures also mean less dilution air is available since a larger amount of air is required for cooling of the flame tube walls. Thus, a greater understanding of the factors influencing the dilution zone performance has been necessary in order to try and achieve the desired temperature profiles in the shortest system length and with a minimum amount of dilution air. At the operating conditions of today's modern engines a 10°C increase in the mean combustor exit temperature can halve the turbine blade life and so extensive research on the behaviour of dilution jets has been conducted, since this is a major factor in determining the life of the turbine stage and the overall performance of the gas turbine engine.

1.2 A General Review of Jet Mixing

In the dilution zone of a combustion chamber a row of jets are injected into a cross-flow formed by the combustion products flowing downstream from the primary and intermediate zones. Now the interaction of a jet, or a row of jets, with a cross-flow has been the subject of intensive investigation for many years since it has important practical implications in a wide variety of areas. The phenomenon occurs, for example, in the discharge of exhaust gases from a chimney, the injection of waste water into a stream, in certain types of film cooling and in VSTOL technology in addition to the mixing processes occurring in combustors where the additional complication of a confined cross-flow arises.
1.2.1 Single jet in a Cross-flow

To establish a flow field within a liner and ensure the correct distribution and mixing of dilution air across the annulus, some knowledge is required of the factors controlling the trajectory of an air jet and its penetration. Numerous studies have been undertaken on a single jet issuing into a cross-flow, the jets being generated by a pipe, nozzle or plenum feed. It is these studies that have therefore been used to indicate the most important parameters that are relevant to the mixing of dilution jets in a combustion chamber.

Many workers have investigated the trajectory of a jet at various operating conditions. For example, Platten and Keffer\textsuperscript{[1]} and Keffer and Baines\textsuperscript{[2]} looked at the trajectory of a jet as defined by its loci of maximum velocity, whilst Abramovich\textsuperscript{[3]} compared the measured trajectory with an empirical equation of the form: \[
\frac{X}{D} = J \left( \frac{Y}{D} \right)^{2.55} \left( \frac{Y}{D} \right) (1 + J)
\]

In addition, several workers including Ramsey and Goldstein\textsuperscript{[4]} and Kamotani and Greber\textsuperscript{[5]} investigated both heated and unheated jets from which the velocity trajectory could be compared with the loci of maximum temperatures. The correlation parameters derived by Kamotani and Greber indicate how the temperature trajectory always falls below the velocity trajectory (Fig.1.3): 
\[
\frac{Y}{D} = 0.89 J^{0.47} \left( \frac{X}{D} \right)^{0.36} \\
\frac{Y}{D} = 0.73 J^{0.52} \left( \frac{\rho_i}{\rho_c} \right)^{0.11} \left( \frac{X}{D} \right)^{0.29}
\]

Furthermore, as indicated by the above equations together with all the other published data, the jet velocity and temperature trajectories are determined mainly by the jet to cross-flow momentum flux ratio (J), although Kamotani and Greber suggested that the temperature trajectory may also have an additional weak dependence on the density ratio. However, as can be seen in their correlation parameters the density ratio appears to the power of 0.11 and so the density effects are relatively small, at least up to temperature differences of 150°C.

Some scatter in the experimental results of the above workers is evident but as outlined by Platten and Keffer\textsuperscript{[1]} this can be explained by the method used to generate the jet for each investigation. For example, the exit velocity profile associated with a plenum fed orifice is different to that of a jet supplied from a pipe or nozzle, and as indicated by Hancock\textsuperscript{[6]} in his aerodynamic review of a jet in a cross-flow the exit conditions are important in determining
Introduction

the subsequent behaviour of a jet. The sensitivity of these effects is indicated by Kamotani and Greber who showed a 12% spread in the velocity trajectories when an ordinary pipe was used to generate the jet instead of a nozzle. The importance of the feed conditions to a jet is also indicated by a series of tests conducted with different injection angles which were designed to simulate geometries such as VSTOL aircraft nozzles or a hole supplied from a feed annulus. Here the approach velocity of the flow results in an effective change in the injection angle and, for example, Platten and Keffer have studied the effect of this in terms of the change in jet trajectory (Fig.1.4). In addition Krausche et. al. studied the effects of the injection angle on the properties of the vortices which are known to develop in the flow field downstream of the injection plane.

All of the above studies are concerned with a single jet being injected into a relatively unconfined cross-flow and very little work has been undertaken with a confined cross-flow geometry such as that found in the dilution zone of a combustion chamber. This is due to the limited application of a single jet issuing into such a configuration, although some data was provided by Stoy and Benhaim in order to assess the accuracy of predicted trajectories derived from theories extended from the unconfined case. This work, however, was related more to the blade impingement cooling schemes that are employed in gas turbine engines. Of greater significance to this investigation though was the work of Kamotani and Greber who compared the velocity distributions of two impinging jets with that of a single jet and opposing wall. Over the range of momentum flux ratios tested (ie. 8 to 32) the trajectories and distributions were virtually identical and so in terms of velocity the plane of symmetry between two opposing jets can be considered equivalent to a wall. It therefore follows that the results obtained from a single jet injected towards an opposite wall can be applied to combustors with directly opposed dilution jets, although for this to be true it is important that the velocities of the two opposing jets be closely matched.

In addition to describing the behaviour of a jet in terms of its trajectory, many workers have studied the velocity and temperature distributions of the cross-section of a jet at various downstream locations. For example Chassaing et. al. measured the velocity profiles in the plane of symmetry of the flow and indicated a similarity law for the axial profiles as close as X/D=0.625 to the injection plane for a velocity ratio of 3.95. At a similar ratio Keffer and Baines also demonstrated that, when characterised by a suitable length and velocity, similarity of the lateral profiles was evident. However, this was only downstream of the jet potential core, the initial region close to the injection plane where the flow is virtually free of shear and the velocity is still that of the exit plane value. Detailed measurements of the jet cross-section and its development in the cross-flow have also been undertaken by several workers including Moussa et. al., Crabb et. al., Lineker and Andreopolous and Rodi, all of which operated at a similar momentum ratio to that of the present investigation. Furthermore, workers such as Fearn and Weston concentrated on the
properties of the vortices which develop downstream of the injection plane and which have a significant effect on jet mixing.

1.2.2 General Flow Characteristics of a Single Jet in a Cross-flow

Initially, work by Abramovich\(^3\) and Keffer and Baines\(^2\) gave a qualitative description of the interactions that occur when a jet is injected into a cross-flow (Fig. 1.5). As the jet of fluid issues into the cross-flow it creates a blockage, and as a consequence the flow immediately ahead of the jet decelerates causing an increase in pressure. Downstream a rarefaction occurs and this, combined with the upstream pressure provides a force that deforms the jet. Because of the intensive mixing of the jet with the cross-flow, a turbulent shear layer rapidly develops around the periphery of the jet and reduces the size of the potential core. The lower momentum fluid in the shear layer at the sides of the jet will, under the influence of the pressure forces, take on a more curved trajectory than that of the higher velocity fluid in the core. This gives rise to the characteristic kidney shaped jet profile that develops downstream of the injection plane. As a result of the action of the deflecting flow the particles of the jet branch out as the jet progresses downstream with the legs of the horseshoe moving further apart, thereby giving rise to circulatory zones which were thought by the early workers to give rise to the formation of a vortex pair.

Improvements in measurement techniques have permitted a large number of workers to conduct detailed quantitative investigations of the flow field and have established the presence of a well defined counter-rotating vortex pair downstream of the injection plane. Chassaing et. al.\(^10\) studied the wake region of a jet and observed the double vortex structures similar to those behind a solid body, but the investigation was more applicable to the emission of effluents into the atmosphere from a chimney. At operating conditions specifically related to the dilution zone of a combustor Crabb et. al.\(^12\) made measurements which encompassed the entire mixing region. Laser Doppler Anemometry was employed in regions where the turbulence intensities were large (X/D<6) with hot wire anemometry being used further downstream, and these results confirmed and quantified the double vortex characteristics of the downstream flow field. Moussa et. al.\(^11\) concentrated on making detailed measurements of the near field flow characteristics close to the injection plane (X/D<1.0), since it was concluded that this region characterises the subsequent behaviour and development of the jet. Contours of mean velocity and vorticity illustrated the formation of the vortex system and it was suggested that, since the jet was formed by a pipe, it is the mean vorticity issuing from this pipe which is reorientated and bundled up into a pair of vortices which are bound to the lee surface of the jet. However, perhaps the most comprehensive investigation and explanation of the vortex flow field downstream of the injection plane was conducted by Andreopoulos and Rodi\(^14\) (Fig. 1.6). Whilst at small momentum flux ratios (J<0.6) the
Introduction

Bound streamwise vorticity originates from the vorticity in the pipe from which the jet is formed, at larger ratios which is more applicable to dilution jets it is the interfacial shear between the jet and cross-flow which is important in generating the vorticity. Streamwise vorticity is directly generated by the high velocity gradient at the sides of the jet which forms the bound vortex system, and this is strengthened by the reorientation of vorticity components perpendicular to the injection wall which are generated by cross-flow fluid moving around the sides of the jet and into the wake region. The experimental findings of Andreopoulos and Rodi also agree with the conclusions reached by Sykes et. al.\textsuperscript{17} whose numerical model indicated that the source of the streamwise vorticity in the vortex pair could be traced to the original streamwise vorticity generated at the sides of the jet. It is these counter-rotating bound vortices which enhance the kidney shaped cross-section of the jet and dominate the downstream flow field, the vortices eventually decaying under the action of turbulent stresses. The dominating effect that the vortex flow field has on the development of the jet is further indicated by Fearn and Weston\textsuperscript{16} where the measured velocity field downstream of the injection plane could be reconstructed using a simple model incorporating a pair of contra-rotating vortices of suitable strength and location.

1.2.3 Prediction Methods

Early methods concentrated on predicting the trajectory of a jet in a cross-flow and involved an integral method approach where the forces acting on an element of the jet column were analysed. For ease of mathematical solution simplifying assumptions had to be introduced concerning the behaviour of the jet, and one of the earliest solutions was derived by Abramovich\textsuperscript{31} using the momentum equation normal to the jet centreline (Fig.1.7). The curvature of the jets axis was obtained by balancing the force due to the the pressure difference at the forward and back surfaces of the jet with the centrifugal body forces. The aerodynamic pressure force is analogous to the drag of a cylinder and was assumed to be proportional to the component of the cross-flow dynamic head that is normal to the axis of the jet. Thus,

\[ F_n = \frac{1}{2} \rho (V_c \sin \theta)^2 C_n b \, dl \]

where \( C_n \) is a drag coefficient and \( b \) is a characteristic width of the jet element over length \( dl \). This pressure force balances the centripetal force which acts on the mass \( dm \) that is associated with this element of the jet i.e.

\[ F_c = \frac{u^2}{R} \, dm = \frac{u^2}{R} \rho A \, dl \]

where \( R \) is the radius of curvature and \( A \) is the cross-sectional area of the jet element.
Thus,

\[ \frac{1}{2} \rho (V_c \sin \theta)^2 C_{n} b \, dl = \frac{u^2}{R} \rho A \, dl \]

Whilst the radius of curvature can be expressed mathematically in terms of \( \theta \) or \( X,Y \) coordinates several additional simplifying assumptions are required to obtain a solution. For example, in the case of Abramovich the width of the jet is assumed to vary linearly with distance along the jet axis so that

\[ b = b_0 + \text{(constant)} \]

whilst the jet velocity \((u)\) is derived by assuming the total momentum of the jet, perpendicular to the undisturbed cross-flow direction, is constant i.e.

\[ \rho u^2 A \sin \theta = \rho u_0^2 A_0 \sin \theta_0 = \text{constant} \]

where \( u_0, A_0 \) and \( \alpha_0 \) refer to the jet exit conditions. However, as outlined by Schetz\[19\], neglecting the variation of the normal momentum means that unrealistically high values of the force coefficient \( C_n \) have to be used in order to obtain satisfactory results. More recent integral method approaches have therefore relaxed this assumption whilst extending the technique to predict the inner structure of the deflected jet. A good example of this is provided by Adler and Baron\[19\] who numerically solved a quasi 3 dimensional integral method to predict the jet cross-section in addition to the jet trajectory. In this case the velocity profile was of the form (Fig.1.8)

\[ u = V_c \cos \theta + Uh \]

where \( U \) is the normalised velocity in the jet direction. The centripetal force on the element of fluid is therefore given by

\[ F_c = \frac{u^2}{R} \, dm = \frac{\rho}{R} \int_A (V_c \cos \theta + Uh)^2 \, dA \]

This can be written in the form

\[ F_c = - \rho \left( I_2 h^2 + 2I_1 h V_c \cos \theta + AV^2_c \cos^2 \theta \right) \frac{d\theta}{dl} \, dl \]

where \( I_1 = \int u \, dA \), \( I_2 = \int u^2 \, dA \) and \( R(-d\theta) = dl \). As in the method adopted by Abramovich, this centripetal force is balanced by a pressure 'drag' force which is a function of the cross-flow velocity component normal to the jet axis (i.e \( 0.5C_n b V_c^2 \sin^2 \theta \, dl \)). However, this method also takes into account the momentum flux associated with the cross-flow which is entrained into the control volume and which has a component \( V_c \sin \theta \). The resulting momentum equation in the direction normal to the centreline is therefore :-
\[ F_e = \frac{1}{2} \rho V_e^2 \sin^2 \theta \cdot C_n b \, dl + \rho E \, dl \cdot (V_e \sin \theta) \]

where the entrainment coefficient, \( E \), is given by:

\[ E = \frac{d(h_i_1)}{dl} + V_e \frac{d(A \cos \theta)}{dl} \]

This equation can therefore be rewritten in the form

\[ \frac{d\theta}{dl} = \frac{\frac{1}{2} V_e^2 \sin^2 \theta \cdot b \cdot C_n + V_e \sin \theta \left( h \frac{dl}{dl} + I_1 \frac{dh}{dl} \right) + V_e^2 \sin \theta \cos \theta \cdot \frac{dA}{dl}}{V_e^2 \sin^2 \theta - V_e^2 A \cos^2 \theta - 2I_1 h \cdot V_e \cos \theta - h^2 I_2} \]

The above equations indicate the more sophisticated approach adopted in recent years in comparison with the earlier integral methods such as that presented by Abramovich. Furthermore, a momentum equation can be written in the jet axis direction which takes into account the change in static pressure across the element of the jet being considered \((dp/dl)\) and the momentum flux associated with the jet and the entrained cross-flow i.e.

\[ A \frac{dp}{dl} + \frac{d}{dl} \int_A \rho u^2 \, dA = \rho E V_e \cos \theta \]

which can be presented in the form

\[ \frac{dh}{dl} = \frac{V_e^2 A \frac{d\theta}{dl} \sin 2\theta - \frac{dI_2}{dl} h^2 - V_e h \left( \frac{dI_1}{dl} \cos \theta - \frac{d\theta}{dl} \cdot 2I_1 \sin \theta \right)}{2I_2 h + I_1 V_e \cos \theta} \]

In order to solve the 2 momentum equations Adler and Baron assumed the entrainment rate \((dA/dl)\) to be a linear combination of a modified straight jet entrainment and the entrainment into a vortex pair. Non-dimensional velocity profiles were also used to permit the determination of \( b, I_1 \) and \( I_2 \) associated with the jet. Furthermore, the shape of the jet cross-section was determined by evenly seeding a finite number of vortices on the instantaneous boundary of the jet and calculating their displacement over a small time period, \( dt \), due to their mutually induced velocities. The new positions of the vortices then describe the new cross-sectional shape of the jet. The complexity of this integral method is typical of modern attempts to predict not only the trajectory of the jet but also its cross-sectional shape. In addition, a further complication arises for the non-isothermal jet case, since in order to predict the temperature distribution it is necessary to include an energy equation in the analysis.

The integral method technique can be used to give a relatively good prediction of the trajectory of a jet issuing into an unconfined cross-flow, and to a more limited extent the shape of the jet cross-section (Fig.1.9). It also indicates the major forces that the jet fluid is
Introduction

subjected to as it issues into the cross-flow. However, the nature of this method means that
the mass of jet fluid is treated as a single element upon which the various forces act. Thus, an
inherent assumption is that all of the velocities within the jet are parallel to the jet axis and so
the internal movement of jet fluid that occurs cannot be modelled. In addition, the vortices
and other flow features which develop downstream of the injection plane and their significant
local effects on jet mixing cannot easily be simulated. The only method of obtaining such a
detailed prediction of the overall flow field and jet mixing characteristics is to obtain a
numerical solution based on the Navier Stokes equations. However, the nature of the
problem means that none of the flow parameter components can be neglected and so a full 3
dimensional solution is therefore required. Furthermore, a large number of grid points is
needed due to the high gradients in the flow field. In most problems the number of grid
points can be minimised by concentrating them in the shear layers that are present in the flow
being studied, but for a jet in a cross-flow these shear layers are curved and their location
unknown so that the grid cannot be optimised using this technique. An accurate solution with
sufficient resolution therefore requires a fine grid of data points in the complete flow field,
although the number of points can be reduced by assuming symmetry about the hole
centre-plane so that the equations need only be solved for one half of the jet. However, this
still results in large amounts of computing time being required in addition to the problems of
obtaining a satisfactory turbulence model for solving the equations. Thus, the majority of
publications have so far concentrated on the accuracy of the solutions obtained rather than
using the model as a method of investigating the flow field and mixing characteristics of a jet,
and the important parameters which influence jet behaviour. However, more recent
publications such as that provided by Sykes et. al.[17] solved the Navier Stokes equations
using a kinetic energy turbulence model and obtained good qualitative and reasonable
quantitative agreement with available experimental data (Fig.1.10). This model was then used
to provide detailed data which gave an insight into the vorticity and its development close to
the injection plane, indicating how the various vorticity components are stretched and
reoriented as the jet is deflected by the cross-flow.

1.2.4 Multiple Jets in a Confined Cross-flow

The limited application of a row of jets issuing into a confined cross-flow means that
the experimental investigations conducted on this geometry are designed to specifically
simulate the mixing processes that occur in the dilution zone of a combustion chamber.
However, in many of these tests the dilution jets were formed from an orifice plate fed by a
plenum and so the feed conditions to the holes were not representative of an operational
combustor. The performance of the dilution jets was usually assessed in terms of the
temperature distributions at a limited number of downstream traverse stations, these being
presented relative to the cross-flow and dilution air temperatures upstream of the injection plane. No attempts were made in these investigations to measure the structure of the jets and the flow features which are responsible for the mixing of the jets and the measured temperature distributions.

A comprehensive study of the mixing of air jets injected into a rectangular duct was conducted by Srinivasan et. al.[20-23] in which the large number of aerodynamic and geometric variables involved in defining the dilution process were investigated. As was the case for the behaviour of a single jet the most significant flow variable for a given geometry was found to be the jet to cross-flow momentum flux ratio ($J$). This is illustrated by the empirical equation developed from this work for the thermal centreline trajectory:

$$
\frac{Y}{D_j} = 0.539 \left( \frac{S}{D_j} \right)^{0.25} \left( \frac{H}{D_j} \right)^{0.14} \left( \frac{X}{D_j} \right)^{0.38} e^{-b}
$$

where

$$
b = 0.091 \left( \frac{X}{H} \right)^{2} \left( \frac{H}{S} \cdot \frac{J}{3.5} \right)^{0.5} \quad \text{and} \quad D_j = D C_d^{0.5}
$$

At operating conditions relating to this investigation, the above equation also indicates the decreased penetration exhibited by a row of jets issuing into a confined cross-flow when compared with the trajectory of a single jet (Fig.1.11), as predicted by Kamotani and Greber[6]. In the series of tests conducted by Srinivasan et. al. a large number of geometrical parameters were varied over a range of momentum ratios (ie. from 6 to 60) and their effect on the downstream temperature distribution assessed. For example, decreasing the hole spacing ($S$) with a constant orifice diameter ($D$) not only reduces jet penetration but also increases lateral uniformity, whilst increasing the orifice diameter at a constant spacing to diameter ratio ($S/D$) improves penetration but increases lateral non-uniformity. A beneficial effect on jet mixing is achieved by reducing the flow area downstream of the injection plane, particularly when it is the injection wall which is responsible for the area convergence, whilst the cross-flow temperature profile upstream of the injection plane also has a significant effect on the downstream distributions. In addition, several types of dilution hole were tested including circular, bluff and slanted slots, a similar investigation also being conducted by Norgren and Humenik[24] who concentrated on holes with chutes on either the feed or cross-flow side of the injection wall. In the latter case the tests also had representative feed conditions since the dilution air was supplied from an annulus although mixing performance was assessed from mean radial profiles only. Both investigations indicated the shape of the dilution ports can have a significant effect on the mixing and trajectory of the jets, the effect of the 45° angled slots studied by Srinivasan et. al. being to laterally shift the temperature centre-plane and alter the shape of the temperature profiles (Fig.1.12). Although no velocity measurements were made this effect was thought to be due to a 'skewed' vortex flow field downstream of each
angled slot which was also thought responsible for rotating the temperature contours about
the slot axis. It is from these extensive investigations that empirical correlations were
developed from which the temperature distributions could be predicted in the dilution zone
for a given set of operating conditions. Some of these results obtained in a rectangular mixing
duct were also confirmed by Sridhari[25] in a circular duct who indicated that for a specific
configuration an increase in jet flow improves the level of mixing up to an optimal value,
after which any further increases causes the temperature distribution to deteriorate as the jets
over penetrate in the mixing duct.

The work outlined above was conducted with single sided injection but both Srinivasan
et. al.[21] and Wittig et. al.[26] investigated opposed jet injection, although in the latter case
performance assessment was only in terms of the mean radial profiles. The results of
Srinivasan et. al. confirmed Kamotani and Greber's observations of a single jet in that the
effect of a plane of symmetry is similar to that of an opposite wall, and this was a primary
assumption in the empirical model that was derived from these results. However, it should be
noted that allowance must be made for the enhanced rate of mixing that occurs due to a
change in the effective duct geometry for the opposed jet configuration ie.

\[
H_{eq}^{top} = \frac{A_{h}^{top}J_{top}^{0.5}}{A_{h}^{top}J_{top}^{0.5} + A_{h}^{bot}J_{bot}^{0.5}}H
\]

where \(H_{eq}\) is the equivalent duct height and the subscripts refer to the top or bottom row
of holes. Furthermore, Wittig et. al. indicated that the plane of symmetry assumption is
limited and results from single sided tests can only be applied to a combustion chamber with
opposed jets if the momentum ratios of the opposed jets are closely matched.

Very few theoretical methods are available for predicting the behaviour of multiple jets
issuing into a confined cross-flow. However, one method of obtaining a solution is to model
a single jet but with modified boundary conditions which simulate both the presence of
adjacent jets and the confining nature of the cross-flow geometry. Difficulties in simulating
these boundary conditions though mean that the integral method technique is not suitable for
the multi jet problem and so, until recently, prediction methods were often based on empirical
correlations such as those presented by Srinivasan et. al. Developments in three dimensional
numerical modelling and more powerful computers for performing the calculations mean that
codes are now becoming available for predicting the mixing of multiple jets in a confined
cross-flow. An example of such a model is outlined by Srinivasan and White[23], and this
was further developed by Reynolds and White[27] to include modelling of the dilution zone in
converging ducts, dual sided injection, reverse flow combustors etc. However, comparison
of the predicted temperature and velocity profiles with that of experimental data shows the
limited accuracy provided by the solution (Fig.1.13). Hence, instead of using such codes to
try and accurately predict the flow field associated with a specific geometry, these codes are
more often used to indicate the general trends and relative effects on jet mixing of changing various aerodynamic and geometric parameters.

1.3 Review of Work Specific to this Investigation

With the exception of that due to Crabb and Whitelaw\cite{28} nearly all of the published work relating to dilution zone performance has concentrated on the mean radial profile, but of equal importance is the consistency of the temperature pattern around the annulus at the exit of a combustion chamber. The discrete fuel injection locations produce variations in the temperature of the flow issuing from the primary zone, and the mixing of each dilution jet with that of the combustion gases produces a further localised change in temperature. Since the length of the dilution zone does not permit complete mixing of a jet with the hot gases a temperature pattern is observed at the exit of a combustion chamber (Fig.1.14), and this should be duplicated as the geometry of the combustor repeats itself around the annulus. The avoidance of variations in these temperature patterns from one sector of the annulus to another is particularly important since this can have a detrimental effect on engine performance and durability of the nozzle guide vanes and turbine blades. However, with the continuing pressure for increased cycle temperatures and combustors of reduced length such distortions of the temperature patterns do occur, often in a random manner and vary in magnitude and position with different combustors built to the same design. Some of these asymmetries may be attributed to a disparity of fuel flow or injector characteristics around the combustor, and there may also be some circumferential variation of airflow issuing from the diffuser. However, even in experiments with carefully controlled and uniform primary zone exit conditions the effect has persisted and this has led to the suspicion that at least some of these distortions are created within the dilution zone.

The work of Shaw and Wilson at Rolls-Royce\cite{29-38} initially involved a parametric study of the variables which influence the dilution process, but later investigations concentrated on the consistency of the temperature pattern around the annulus downstream of the injection plane. The majority of these tests were conducted on a Viper annular combustor, the front of which had been removed and within which no combustion took place. Instead, a second combustor mounted approximately 2.5 metres upstream of the working section was used to supply hot gas and this, together with numerous gauzes and flow control devices, ensured extremely uniform primary zone exit conditions in the working section. However, even with these upstream conditions it was only with a plenum feed to the dilution holes could variations in the temperature patterns around the annulus be minimised. When casings were introduced to simulate the annuli which supply air to the dilution holes the irregularities in the temperature distribution increased, being at a maximum for the smallest feed annulus
height tested. Furthermore, large variations in the flow angles at the centre of each hole were recorded by yaw meters and 5 hole probes. In an attempt to reduce the temperature irregularities longitudinal ‘splitter’ plates were mounted across each dilution hole together with other flow control devices, but these produced no improvement in the consistency of the temperature pattern. Although no measurements were made on the behaviour and mixing of the jets in the dilution annulus, it was thought that the temperature irregularities could be due to the impingement of jets which have insufficient directional bias or a lack of symmetry between the dilution jets. It is this latter effect which has been investigated in the Airflow Laboratory at Loughborough University, the experimental programme concentrating on the consistency of the temperature patterns produced by the localised mixing of a row of heated jets issuing into a confined cross-flow.

A test facility was initially built at Loughborough University in 1979 to investigate the effect of manufacturing inaccuracies on dilution jet mixing, since the tests at Rolls Royce had indicated that slight variations of the flow distribution or height of the dilution hole feed annulus could lead to irregularities in the combustor exit temperature distribution. In addition, tests at Imperial college[28] on a 2-D rig incorporating single sided injection had indicated that slight circumferential misalignment of a dilution hole, typical of that which may occur in a combustor, results in a large asymmetry of the flow field and may lead to temperature irregularities at the exit plane of a combustor. A fully annular test facility incorporating single sided injection was therefore built at Loughborough, with 16 heated dilution jets issuing into a confined cross-flow. The consistency of the temperature pattern and mixing of the jets around the annulus being assessed from thermocouple measurements made downstream of the injection plane.

Initial tests were conducted with the 16 dilution jets being fed from a plenum and, as was also found in tests at Rolls-Royce, this produced a very consistent temperature pattern around the dilution annulus and provided a datum for the purposes of comparison with subsequent results. Moys and Stevens[39] investigated the effects of laterally offsetting one of the dilution holes by up to 0.75 diameters, but only local asymmetry was found in the downstream temperature distribution with the global annulus condition remaining stable and so the disturbances did not propagate beyond the sector of origin.

At this stage it was suggested that the most likely causes of temperature irregularities in combustion chambers were variations in the velocity and pressure of the air fed to the dilution holes. The rig was therefore modified by Malik and Hyde[40] so that the dilution holes were supplied from a feed annulus, the approach flow of finite velocity simulating more closely the conditions found in combustion chambers. A series of ad-hoc tests were then carried out aimed at determining what forms of perturbation promote inconsistencies in the temperature pattern around the annulus. In an attempt to simulate the flow downstream of a burner feed arm Bailey[41] introduced a wake flow in the feed annulus upstream of a dilution hole whilst
Wilson\cite{42} investigated the effects of a local reduction in the height of the feed annulus. However, the results from these tests proved difficult to analyse due to the high level of temperature inconsistencies encountered when the dilution holes are supplied from a feed annulus, so that the changes made by Bailey and Wilson could not be detected in terms of the consistency of the temperature patterns. Attempts to isolate the influence of specific hole geometry from approach annuli conditions were also unsuccessful due to an inability to repeat the test data when the rig was restored to the datum condition.

1.4 Objectives and Scope of the Present Investigation

The aim of this investigation is to initially undertake a major redesign of the test facility to improve the uniformity of the feed conditions to the dilution holes. This should eliminate any contribution to the variations in the temperature patterns around the annulus that are associated with local upstream flow conditions whilst also improving the repeatability of the test data. Despite relatively large localised geometric variations being introduced by Bailey and Wilson no significant change was observed in the level of the temperature irregularities and so a fundamental investigation is therefore to be conducted with the rig in its datum configuration. This is because large variations in the temperature patterns around the dilution annulus are observed merely when the row of dilution holes are supplied with air from a feed annulus. The magnitude of the temperature irregularities are to be assessed and individual dilution jets then studied to observe their structure and mixing characteristics in order to discover why their associated temperature patterns vary with respect to other jets. With the causes of the problem known a solution can be offered which should improve the overall regularity of the temperature patterns around the annulus.

Although the geometry of the test facility is based upon an annular combustion system, it is thought that the structure and mixing characteristics of the dilution jets and the findings of the investigation can be applied to both tubo-annular and tubular combustor designs.
Chapter 2: TEST FACILITY

2.1 Description of the test facility 17

2.2 Instrumentation 19

2.3 Rig traversing 22
   2.3.1 Complete annular surveys 22
   2.3.2 Sector surveys 23

2.4 Rig operating conditions 24
An investigation into the circumferential regularity of the temperature pattern around the annulus of a dilution zone requires a fully annular test facility. However, rather than construct a test rig incorporating a dual sided dilution system such as that found in an annular combustor, a single sided system was designed since both Kamotani and Greber\cite{9} and Srinivasan et. al.\cite{21} have shown that a plane of symmetry exists between opposed jets which can be considered as equivalent to a wall. This significantly reduces the size and complexity of the test rig and permits easier access for instrumentation close to the injection plane. Due to the constructional materials, instrumentation and running costs it is not practical or desirable to operate at temperature levels that exist in a modern combustion chamber, though some means of assessing the mixing performance of the jets is required. A relatively small temperature difference of 44°C is therefore introduced between the cross-flow (55°C) and dilution air streams (11°C) and for practical reasons heated jets are injected into a relatively cold cross-flow, this being the opposite situation to that found in a combustion chamber. However, by selecting suitable operating conditions the temperature difference gives the required information on the mixing of jet fluid around the annulus which is thought sufficient to fulfil the objectives of this research programme. Furthermore, it is important to note that no heat is applied to the dilution air when velocity measurements are being made in the mixing annulus using a 5 hole probe. However, by maintaining the same momentum flux ratio ($J$) the behaviour of the unheated jets can be made to simulate those of the heated jets upon which the temperature measurements are made.

2.1 Description of the Test Facility.

The test facility (Fig.2.1(a)) is comprised of 3 vertically mounted concentric Perspex tubes 9.5mm thick, the space between the inner and centre casings forming the dilution hole feed annulus of 35.8mm (1.41D) height, and the space between the centre and outer casings forming the cross-flow annulus of height 76.2mm (3D). The concentricity of these tubes has been measured to be within 0.15% of the annulus heights. The dimensions of these annuli are not based on a specific combustor, but reflect the average geometry which was obtained from a survey of military annular combustion chambers which operate in the Olympus, Pegasus, RB199 and M45H gas turbine engines\cite{43}. In addition, the length of the tubes are designed to ensure a suitable boundary layer thickness in each annuli upstream of the dilution holes and the required length for mixing of the flow downstream of the injection
plane. For ease of construction each tube consists of a number of sections, and by mounting the test facility in the vertical plane the influence of buoyancy on the penetration of the dilution air is the same for all jets. Furthermore, this also eliminates the need for support struts between the casings and so avoids the presence of wakes in the cross-flow and feed annuli which could affect the mixing characteristics of specific jets.

A 10kW heater is supplied with air drawn from the laboratory by a centrifugal fan (Fig.2.1(b)) which is driven by an 11kW electric motor, the speed of the fan being manually regulated by varying the resistance in the rotor circuit of the A.C. slip ring motor. For a given mass flow the rise in temperature of the air passing through the heater can also be controlled by varying the resistance in the associated electrical circuit. The heated air then passes into a large 254mm diameter pipe where the relatively low air speed (≈ 5m/s) ensures that any local variations in air temperature are eliminated and a uniform temperature distribution is obtained prior to the flow passing into the dilution hole feed annulus via an annular nozzle. Air is drawn from atmosphere by a second fan into a 19.6m³ plenum chamber and then passes into the cross-flow annulus via a bell shaped inlet flare, air filter and a honeycomb flow straightener. At a distance of approximately 1.5m downstream of the inlet flare the heated dilution air is injected into the cross-flow and the 2 air streams mix prior to being dumped into a 2.2m³ plenum chamber at the base of the test rig. The second fan then expels the mixed air to atmosphere through an exhaust duct system.

At the injection plane (Fig.2.1(c)) the array of dilution jets are formed by 16 equi-spaced holes of 25.4mm diameter located in the centre casing and which have been numbered 1 to 16 in an anti clockwise direction when viewed from the base of the rig. In the standard configuration the dilution holes are of conventional plunged design \((R_p/D = 0.375)\) and are set at a spacing to hole diameter ratio \((S/D)\) of 2.75 and a mixing duct height to hole diameter ratio \((H/D)\) of 3. So that the holes are virtually identical with a similar plunging surface finish, each hole is manufactured separately from flat Perspex sheet and then inserted into the cylindrical casing (Fig.2.2). The holes are blended in and contoured to provide a smooth surface with no discontinuities on the inside of the casing, whilst a plasticene fillet is added to obtain a flat injection wall surface. Using this technique no physical variation could be measured around the casing from one hole to another.

If air is allowed to flow without restriction into the space downstream of the dilution holes disturbances can arise that cause air to recirculate upstream in the feed annulus in an intermittent and random manner as outlined by Lefebvre\(^{[44]}\), thereby causing flow instabilities which vary in an irregular manner with time. As adopted in many combustion chambers the area of the feed annulus is therefore reduced, in this case by an annular wedge on the inner casing adjacent to the dilution holes and which reduces the passage height from 35.8mm \((1.41D)\) to 10.4mm \((0.41D)\). In this way an excessive rise in pressure across the face of the holes due to diffusion is prevented and the instabilities minimised. In the design of
the test facility some allowance is made for the simulation of flow used for flame tube and turbine blade cooling that normally passes down the annulus between the liner and combustor casing. A circumferential slot 3mm high and located 190mm downstream of the injection plane is designed to allow 6% of the flow in the feed annulus to continue downstream of the dilution holes and then be discharged into the cross-flow. However, as shown by Wilson[42] this design proved unsatisfactory with cross-flow tending to enter the dilution hole feed annulus through the slot setting up unrepresentative recirculating flows. Thus, in the work presented here the slot is closed and no attempt is made to simulate the flame tube and turbine blade cooling air.

Comparison of the test rig in its present form with that of the configuration tested by Bailey[41] and Wilson[42] (Fig.2.3) illustrates the considerable modifications undertaken to change the way in which air is supplied to the dilution hole feed annulus. The air is no longer required to turn through 180° but accelerates smoothly into the annulus from the pipe sited downstream of the heater. In addition, cross-flow air is not drawn from the laboratory but from a large plenum mounted above the rig which is connected directly to atmosphere. In this way a more uniform cross-flow temperature is achieved during each test since the air in the laboratory can vary in a random manner depending on the opening of doors, vents etc.

2.2 Instrumentation

The instrumentation used on the test facility is designed to provide detailed mean temperature and velocity information on the mixing of the jets in the cross-flow.

Mean temperatures are obtained using type K Chromel/Alumel thermocouples which, as outlined by MacDonald[45], have a relatively high thermoelectric output, low thermal conductivity, are tough and durable and have an operating range of -184°C to +1350°C. Each thermocouple is manufactured from 0.3mm diameter insulated wire and to ensure a fast response and minimum measuring volume the thermocouple junction is made as small as possible and is not shielded from the flow. For the temperature range associated with the present application a maximum error of ±0.15°C (BS4937) can occur due to variations in the characteristics of the Chromel and Alumel wires. A calibration in still air is therefore performed which ensures all the thermocouples used on the test facility measure to within ±0.15°C of a reference temperature. To permit accurate positioning in the flow field each thermocouple junction and associated wires are mounted in stainless steel hypodermic tubing with a nominal outside diameter of 3.3mm, each probe being either straight or cranked through 90° depending on if measurements are to be made in planes perpendicular or parallel to the injection plane (Fig.2.4). Since each probe is stationary relative to the moving air flow it should be remembered that the signal recorded by the system is a function of the air
velocity as well as the static temperature (see section 3.5.1). In addition to the probes mounted in the cross-flow annulus, 2 further thermocouples are positioned upstream of the injection plane and monitor the rig operating temperatures. One thermocouple is used to record the temperature of the air in the plenum chamber supplying the cross-flow annulus whilst a second thermocouple monitors the dilution air temperature downstream of the heater.

All thermocouples are connected to microprocessor controlled Comark digital thermometers which provide a reference cold junction and automatic calibration of the signal produced by the Chromel Alumel wires. Each Comark has a 10 channel input with a single D.C. analogue output channel which provides a linear 0 to 1 volt signal corresponding to a temperature range of 0°C to 100°C. A resolution of 0.1°C is quoted by the manufacturers and a manual switch is used to select which thermocouple input channel provides the output signal. To improve the signal to noise ratio the signal from each Comark thermometer is fed through a low pass R-C filter with a cut off frequency of 1kHz. This eliminates any high frequencies superimposed on the signal associated with the internal electronics of each Comark. The response time of each Comark instrument is in the order of 0.5 seconds whilst the small size of each thermocouple junction results in the sensor signal reaching its output value in approximately 1 millisecond. Some allowance must be made for the effect of the low pass R-C filters and so a settling time of 1 second or more is allowed for the complete system, after movement of the instrumentation, before the temperature measurements are sampled by the computer.

Miniature five hole pressure probes are used to provide information on flow angles, velocities, total and static pressures at various locations in the isothermal flow field produced when no heat is supplied to the dilution air. Each pressure probe consists of a cluster of five tubes 0.25mm bore, giving an overall diameter of approximately 1.73mm and are used in a non-nulled mode. This reduces testing time since nulling of a probe at each data point is not required and a much simpler traverse system can also be employed. A detailed description of the theory and calibration procedure is presented by Wray[47,49] to which reference should be made for further information since it is from this work that the present method has been adopted. However, a brief summary of the technique is outlined in Appendix 2. Basically each of the 5 tubes registers a pressure which is a function of the flow direction relative to the probe and the total and static pressures. Each probe prior to being used on the test facility is calibrated by being placed in a flow of known dynamic pressure for a range of pitch and yaw angles. Non dimensional parameters are then calculated and stored on the computer for each flow condition which are based on the differences in pressure between the 5 tubes. The flow direction, total and dynamic pressures can then be obtained when the probe is placed in the flow field of interest by calculating these parameters from the pressure readings and referring to the calibration data. These calculations are performed by off-line software during which corrections are made for the finite size of the probe and variations in the rig operating
Test Facility

conditions. Since it is the dynamic head \((0.5pV^2)\) which is calculated from the 5 hole probe pressure data the velocity measurements are limited to a flow field where the density \((p)\) is of known value. Hence, as already stated the velocity measurements are performed on unheated jets in an isothermal flow field where the density is of known value and assumed constant across the mixing annulus.

Each five hole probe is mounted in the same 3.3mm diameter hypodermic tubing that is used for the thermocouples so that similar traverse mechanisms can be used for the different types of instrumentation. The geometry of the probe tip means that each probe can only can only be accurately calibrated up to approximately \(\pm 36^\circ\) in pitch and yaw relative to the local flow direction. Thus, probes with different geometries have been used (Fig.2.5) in an attempt to obtain as many measurements as possible in the highly 3-dimensional flow field close to the injection plane. Pressures sensed by each hole are measured using sets of Furness pressure transducers which produce an analogue output voltage proportional to the pressure measured. Transducers available include those with input ranges of \(\pm 500\text{mm H}_2\text{O}\), \(\pm 100\text{mm H}_2\text{O}\) and \(\pm 25\text{mm H}_2\text{O}\) which produce a linear D.C. output voltage of \(\pm 1\) volt. Low pass R-C filters with a cut off frequency of \(10\text{kHz}\) are installed on the output from each transducer in order to eliminate high oscillator frequencies which are superimposed on the transducer signals. The time response of the system is determined by the length and bore of the pressure tubing used for each probe and upon movement of the instrumentation an experimentally determined settling time of 10 seconds is adopted prior to the pressures being recorded by the computer.

In addition to the pressure instrumentation directly used in measuring the behaviour of the dilution jets further pressure probes are available for use on the test facility. Pitot probes are mounted at mid-height in the cross-flow annulus, 6 hole diameters upstream of the injection plane and also in the exit plane of several dilution holes. These are used in conjunction with numerous static pressure tapings on the wall of the outer casing to provide information on the rig operating conditions. Rakes of pitot probes (Fig.2.6) located at 16.5 and 3.5 hole diameters upstream of the injection plane in the feed annulus allow the velocity profile of the flow approaching the dilution holes to be measured. These instrumentation probes are used in conjunction either with the aforementioned pressure transducers or micromanometers in order to obtain a pressure reading.

Pressure and temperature data is collected by computer programs which are stored in the memory of an LSI 11/23 microcomputer. All data is based on measurements time averaged over a period of 5 seconds after allowance has been made for the readings to settle following movement of the instrumentation. Signals are supplied to a 12 bit analogue to digital (ADC) convertor taking 250 samples at intervals of 20 milliseconds. The pressure and temperature data, along with operating conditions and positional information, is temporarily stored in the memory of the LSI prior to being transferred to a PDP 11/34 minicomputer.
where it is logged onto hard disk. All subsequent off-line data processing and analysis is conducted on the minicomputer using a suite of computer programs.

2.3 Rig Traversing

The instrumentation used to investigate the mixing of the dilution jets is mounted in traversing mechanisms attached to the outer casing of the test facility. Although the centre tube is mounted on a fixed pedestal, the outer casing with the instrumentation can be rotated and by this means the measurement probes can be positioned at any circumferential location around the dilution annulus. The positional accuracy is estimated to be within 0.05° of the required value which corresponds to a circumferential distance on the injection wall of 0.006D.

2.3.1 Complete Annular Surveys

The work conducted by Bailey[41] and Wilson[42] was limited to these types of tests in which 16 thermocouples are used to map the entire flow field around the dilution annulus (Fig.2.7). Two wooden spacers of height 25.4mm and 50.8mm are incorporated in the outer casing and provide the means by which the vertical position of the entire instrumentation section can be varied with respect to the dilution holes. Measurements can therefore be made in planes 1,2,3 or 4 hole diameters downstream of the injection plane. With the downstream location fixed for a given test, each thermocouple is manually adjusted with respect to a datum position to provide radial location in the cross-flow annulus (Fig.2.8). When positioned at the same radius the outer casing is then rotated so that each thermocouple is simultaneously traversed over a spacing equal to 2.75 hole diameters in increments of 2.5°. The radial position of each thermocouple can then be adjusted and the process repeated so that area traverses are performed in planes perpendicular to the injection wall.

The advantage of this system is that each time the temperatures are surveyed the 16 thermocouples are located at similar positions relative to each dilution hole and any recorded asymmetry is logged at virtually the same operating conditions. Unfortunately the temperatures cannot be logged simultaneously due to the single output channel of each Comark thermometer, so at any location it therefore takes approximately 120 seconds whilst each of the 16 temperatures are recorded along with the cross-flow and dilution air temperatures upstream of the injection plane. However, it is assumed there is negligible variation of the operating conditions during this time interval and so using this technique the overall regularity of the temperature pattern around the dilution annulus can be quantified and assessed.
One disadvantage of this system is that the number of thermocouples and the time required for the tests to be completed limits the number of temperature readings that can be recorded. Thus, associated with any jet are 90 data points, with 9 temperatures being recorded at 10 radii across the annulus. This density of data points is not sufficient to permit a detailed investigation of the behaviour and mixing characteristics of individual dilution jets. So although irregularities in the temperature patterns around the annulus can therefore be quantified, the fundamental reasons as to why these variations occur cannot be deduced. A further disadvantage of surveying the complete annulus is the high experimental errors associated with the measured temperatures. At any location there will be some variation in measured temperatures due to the different characteristics of each thermocouple as outlined in the previous section. Furthermore, since the instrumentation is manually adjusted errors can occur due to variations in the position of each thermocouple relative to the dilution hole. This can lead to errors which become significant in regions of high temperature gradients such as around the periphery of the hot core of each jet.

2.3.2 Sector Surveys

Measurements obtained from both thermocouples and 5 hole pressure probes can be used to define the structure and mixing characteristics of individual dilution jets and relate this to variations in the temperature patterns around the annulus (Fig. 2.9(a) and (b)). Manual traverse mechanisms machined from Brass and externally mounted on the outer casing provide accurate radial positioning for both types of instrumentation whilst the wooden spacers permit measurements to be made at 1, 2, 3 or 4 hole diameters downstream of the injection plane. A probe is inserted into the dilution annulus through one of several small apertures in the outer casing and is then mounted in the traversing mechanism where it can be moved in a translational manner by rotation of a positioning collar and screw thread similar to that of a micrometer type mechanism. To eliminate any free play inherent in the mechanism, the back-lash is taken out by only driving the probe in one direction during a test. Thus, the accuracy of positioning instrumentation in the cross-flow is therefore only limited by setting up errors which have been estimated to be approximately ±0.1mm (0.004D). Measurements are recorded at intervals of 0.5° by rotation of the outer casing so that 45 data points are logged at each radius. This is repeated at up to 28 radii so that as many as 1260 data points are recorded per jet in planes perpendicular to the injection wall. This large number of data points is necessary due to the steep velocity and temperature gradients and small flow features that are present within the flow field. However, although a detailed mean flow pattern is obtained for a segment of the annulus, the limitations of rig running time and microcomputer storage capacity mean that only a single jet can be investigated per test.

As outlined by Moussa et. al.[11] it is the near field region close to the injection plane
which characterises the subsequent behaviour and development of a jet. It is therefore necessary to make measurements close to the dilution holes \((X/D<1.0)\) both in planes parallel and perpendicular to the injection wall. This is achieved by mounting a micrometer type mechanism on the test facility which controls the vertical position of a second radial traverse mechanism in which the probe is mounted. A small slot in the outer casing is sized just large enough to permit the required insertion and movement of the probe in the dilution annulus whilst ensuring that the leakage of air is kept to a minimum. These traverse mechanisms in conjunction with the circumferential movement of the outer casing permit positioning of the instrumentation in any plane close to the dilution holes \((X/D<1.5)\). For measurements close to the injection plane the data is collected at nominal circumferential and radial spacings of \(0.06D\) and \(0.1D\) respectively in planes perpendicular to the centre casing. Data recorded in planes parallel to the centre casing is done so at a nominal spacing of \(0.06D\) in both the circumferential and streamwise directions. The setting up of the probes is assisted by marking the centreline \((p=0.0')\) of each dilution hole on the centre casing.

An advantage of investigating a single jet per test is that very accurate positioning of the instrumentation can be achieved in the dilution annulus. Furthermore, the use of 3 Comark thermometers during temperature tests permit the reference cross-flow and jet conditions to be logged simultaneously with that of the reading in the dilution annulus. Such tests can also be repeated, downstream of several different dilution holes, so that a temperature distribution can be built up for a sector of the annulus containing a number of jets. Since the same instrumentation and positioning equipment can be used for each jet a very accurate assessment can be made of the regularity of the flow patterns around the sector although the operating conditions must be precisely monitored and maintained from one test to another.

2.4 Rig Operating Conditions

The most important factor controlling the behaviour and mixing of the dilution jets downstream of the injection plane is the jet to cross-flow momentum flux ratio \((J)\). Although it is not practical to operate at the temperature levels found in the dilution zone of a combustion chamber the behaviour of the dilution jets can be imitated by running the test rig at a similar momentum flux ratio. The test facility is therefore operated at a jet to cross-flow momentum ratio of 4, since this reflects the operating momentum flux ratios for the combustors upon which the geometry of the test rig is based\[^{[43]}\].

The dilution air is only heated in tests where thermocouple measurements are made, since to accurately calculate the velocity field the density of the flow downstream of the injection plane must be known at every traverse location. With no heat introduced the density of both air-streams is assumed constant and equal to the atmospheric value but the test facility
is operated at the same momentum flux ratio of 4. Thus, the momentum of the jets and their behaviour in the mixing annulus are the same in both heated and unheated tests allowing comparisons to be made of the flow patterns indicated by both types of measurements.

The momentum flux ratio is calculated from measurements of the dynamic head of the flow issuing through the dilution holes \( (0.5p_jV_j^2) \) and in the cross-flow annulus \( (0.5p_cV_c^2) \). The ratio of these values represents the momentum flux ratio \( (p_jV_j^2/p_cV_c^2) \) and takes into account any variations in temperature and pressure between the air streams. It should be noted that in order to measure the cross-flow and dilution jet dynamic heads it is assumed the static pressure is constant across the annulus formed by the outer and centre casings. A static pressure tapping in the outer casing, 6 hole diameters upstream of the injection plane, together with a pitot probe midway between the cross-flow annulus casings permit measurement of the cross-flow dynamic head. A pitot probe mounted in the exit plane of a dilution hole \( (X/D=-0.2, \varphi^*=0) \) measures the total pressure of the jet flow whilst a tapping in the outer casing, opposite the dilution hole, allows measurement of the static pressure at the injection plane. By mounting the pitot probe in the same dilution hole for every test, the dynamic head used for reference purposes is always associated with the same jet. The momentum flux ratio is checked at 15 to 20 minute intervals throughout a test and adjustments are made where necessary to ensure the ratio is maintained within ±0.05 of the nominal value of 4.0.

In addition to maintaining a constant momentum flux ratio, the test facility is also operated to maintain a jet velocity \( (V_j) \) of approximately 29.5m/s (=50mm H\(_2\)O) and, where applicable, cross-flow and dilution air temperatures of 11°C and 55°C respectively. All these quantities may vary from their nominal values and corrections are made in the analysis of the data to account for this.

When commencing a test, the rig is started half an hour or more before any data is logged and, if required, the heater switched on to allow the air streams and rig casings to reach a stable operating condition. This also permits an adequate warm up time for the pressure transducers, Comark thermometers and other electrical equipment to be used during the test. Both the jet and dilution air speeds are varied by 2 motor controllers which allow the operating conditions to be approximately obtained to within 1mm H\(_2\)O of the required dynamic heads. A throttle at inlet to the dilution air fan then permits a fine control of the jet mass-flow thereby allowing the accurate setting and maintaining of the reference momentum flux ratio. The power output from the heater can also be varied manually and a temperature difference of approximately 44°C is maintained between the cross-flow and dilution air streams for tests involving temperature measurements.
Chapter 3: DATA ANALYSIS AND PERFORMANCE PARAMETERS

3.1 Data reduction 27

3.2 Performance parameters 31

3.3 Vorticity definitions 36

3.4 Presentation of data 37

3.5 Estimation of experimental errors 38
  3.5.1 Temperature measurements 39
  3.5.2 Velocity measurements 41
3.1 Data reduction.

To accurately assess the mixing performance of the dilution jets from thermocouple measurements made in the cross-flow annulus it is necessary to consider the operating conditions at which the temperatures were recorded. Because of changes in atmospheric conditions from one test to another, substantial variations can occur in the temperature of the cross-flow and dilution air upstream of the injection plane. This will influence the values measured by the downstream thermocouples in the cross-flow annulus and so it is necessary to correct all these values to a common set of operating conditions. This correction procedure assumes that the value of the local temperature measured at a specific location in the mixing annulus is determined by the amount of cross-flow and jet fluid contained in that local control volume (Fig. 3.1). Using the steady flow energy equation and assuming that downstream of the injection plane no work input or heat exchange with the surrounding casings takes place and that potential and kinetic energy effects can be ignored then:

\[ m' h = m'_j h_j + m'_c h_c \]

where the total mass flow through the local control volume \( m' \) consists of jet fluid \( m'_j \) with upstream enthalpy \( h_j \) and cross-flow fluid \( m'_c \) with upstream enthalpy \( h_c \). Thus

\[ m' = m'_j + m'_c \]

and using the general definition for enthalpy

\[ \delta h = C_p T \]

then

\[ m'_T_{\text{meas}} = m'_j T'_{j_{\text{meas}}} + m'_c T'_{c_{\text{meas}}} \]

or

\[ m'_T_{\text{meas}} = m'_j T'_{j_{\text{meas}}} + (m' - m'_j) T'_{c_{\text{meas}}} \]

This can be rearranged to give:

\[ \frac{T'_{\text{meas}} - T'_{c_{\text{meas}}}}{T'_{j_{\text{meas}}}} = m'_j \]

\[ m' \]
At a constant momentum flux ratio the proportion of jet to total fluid at any local region in the mixing annulus is constant. Thus, the analysis of data assumes that the non-dimensional temperature ($\theta$) remains constant at any location in the dilution annulus.

$$\theta = \frac{T_{\text{meas}} - T_{\text{c_meas}}}{T_{\text{i_meas}} - T_{\text{c_meas}}}$$  \hspace{1cm} (3.1.1)

It therefore follows that if $\theta$ equals one the local temperature in the dilution annulus is that of the jet temperature and a value of zero means the local temperature equals that of the cross-flow.

Alternatively, the measured values can be corrected to a standard set of operating temperatures using the assumption that the non-dimensional temperature is constant at a specific location. Thus, for ease of understanding certain equations and results may be expressed in terms of corrected temperatures ($T_0$) as well as in the non-dimensional ($\theta$) form. This also gives the reader an indication of the magnitude of the effects measured relative to the limitations of the instrumentation system. Hence

$$\theta = \frac{T_{\text{meas}} - T_{\text{c_meas}}}{T_{\text{i_meas}} - T_{\text{c_meas}}} = \frac{T_0 - T_{\text{c}}}{T_{\text{i}} - T_{\text{c}}}$$

where $T_i$ and $T_c$ are reference jet and cross-flow temperatures.

Thus, corrected temperature $T = \left(\frac{T_{\text{meas}} - T_{\text{c_meas}}}{T_{\text{i_meas}} - T_{\text{c_meas}}}\right) \left(T_{\text{i}} - T_{\text{c}}\right) + T_{\text{c}}$ \hspace{1cm} (3.1.2)

Where applicable, all temperature measurements are corrected to a reference jet temperature ($T_i$) of 55°C and a reference cross-flow temperature ($T_c$) of 11°C. A corrected value ($T_0$) therefore represents the temperature that would have been recorded at that location had the test facility been operating with jet and cross-flow temperatures of 55°C and 11°C respectively.

The above non-dimensional temperature technique of data reduction permits a comparison of the results from tests conducted on different days at different operating temperatures, whilst also accounting for variations in the temperature levels that may occur during a single test. In addition to the present investigation this method has been widely and successfully used by many workers in the field of jet mixing studies.

The pressure data obtained from tests involving measurements with 5 hole probes also needs correcting since, although the test facility always operates at a momentum flux ratio of 4.0, significant variations can occur in the mass flow passing through the rig and also in atmospheric conditions. All the pressure measurements obtained from the 5 hole probes are
therefore corrected and, in a procedure similar to that recommended by Wray\cite{47}, measured pressures are referenced to the static value at a reference plane (Fig. 3.2). In such a configuration it can be shown that the difference between the pressure sensed by any tube and the reference static scales linearly with an inlet reference dynamic pressure, but only if the reference plane Mach number is maintained constant i.e.

\[
\frac{(p - p_{ref})_{meas}}{q_{meas}} = \frac{(p - p_{ref})_{std}}{q_{std}} = \text{constant}
\]

where the subscripts meas and std refer to the measured value and the value that would be obtained on a standard day when operating at the correct Mach number. This equation can be written as

\[
(p - p_{ref})_{std} = (p - p_{ref})_{meas} \frac{p_{std}}{p_{meas}} \frac{t_{meas}}{t_{std}} \frac{u_{std}^2}{u_{meas}^2}
\]

At a constant reference plane Mach number (=\(u/(\gamma R t)^{0.5}\))

\[
\frac{u_{std}}{0.5} = \frac{u_{meas}}{0.5} = \text{constant}
\]

Hence

\[
(p - p_{ref})_{std} = (p - p_{ref})_{meas} \frac{p_{std}}{p_{meas}}
\]

Thus, the speed of an experimental test facility can normally be controlled to maintain a constant \(u/0.5\) value at the reference plane, and this therefore takes into account variations in atmospheric temperature (\(t_{meas}\)) and avoids fluctuations in the mass flow rate. The ratio \((p_{std}/p_{meas})\) applied to the measured value \((p - p_{ref})\) corrects for variations in atmospheric pressure from that of the standard day. It is therefore possible to correct all the measured data to values that would have been recorded had the tests been performed on a standard (ISA) day. However, in the present investigation the primary operating parameter maintained during a test is the jet to cross-flow momentum flux ratio \((J)\). This, together with the finite number of speed settings of the cross-flow fan means that it is impractical to also operate at a constant reference plane Mach number \((u/0.5)\). Instead, it is assumed that during any test the air density remains constant and corrections are only applied to account for variations in mass flow due to changes in the operating speeds of the fans. Thus, all pressure measurements are corrected using an equation of the form

\[
(p - p_{ref}) = (p - p_{ref})_{meas} \frac{q_{ref}}{q_{meas}}
\]

Although the correction procedure accounts for changes in operating speeds, it does not allow for variations in atmospheric density. Thus, pressures are corrected to a reference
Dynamic head \(q_{\text{ref}}\) but this does not mean that the corrected values are those which would have been recorded on a standard (ISA) day. Since velocities are calculated from a dynamic head \((0.5pu^2)\) it therefore follows that variations in velocity will occur from one test to another due to changes in atmospheric conditions \((\text{ie.p})\). However, such effects can be taken account of when presenting the data by non-dimensionalising the measured velocities with respect to the jet velocity \((V_j)\) recorded on the day of the test.

Basically two inlet dynamic pressures can be used for the correction process, namely the cross-flow dynamic head and the dynamic head of the flow issuing from the dilution holes. All pressure measurements are conducted downstream of the injection plane and monitor the development of the dilution jets, and so it is therefore assumed that the difference in the pressures recorded from that of the reference static is proportional to the dynamic pressure of the jet fluid issuing from the dilution holes. The pitot probe mounted in the exit plane of a dilution hole is therefore used to constantly monitor the dynamic pressure of the jet flow and used to correct the 5 hole probe readings. By physically connecting each of the 5 tubes to one side of the pressure transducers and the injection plane static pressure tapping to the other, all pressures are automatically obtained relative to the desired datum. The difference between the injection plane static pressure and the pitot probe provides the jet dynamic pressure \((q_j)\) permitting the measurements to be corrected using the formula

\[
(p - p_{\text{ref}})_{\text{corrected}} = (p - p_{\text{ref}})_{\text{meas}} \frac{q_{j\text{ref}}}{q_{j\text{meas}}} \quad (3.1.3)
\]

All measurements are corrected to a reference jet dynamic pressure \((q_{j\text{ref}})\) of 50 mm H\(_2\)O.

When a 5 hole probe is used in a highly sheared flow then the probe readings will be inaccurate simply because of the finite distance between the measurement holes on the probe tip. One way to solve this problem is to make small movements to the probe in order that all the holes in turn could be placed at exactly the same point in the flow field. However, this would incur a five fold increase in testing time and require added sophistication to the traversing mechanism. An alternative technique adopted here is to perform an area traverse with the probe in one position only for each data point, and then to perform a cubic spline interpolation in both the radial and circumferential directions on the individual probe hole readings in order to collapse the pressures for all 5 holes onto one point. This procedure lends itself well to computer manipulation and is carried out after the pressure measurements have been corrected for changes in operating mass flows using equation \((3.1.3)\).

It has already been stated that the five hole probe measurements are conducted on unheated jets but with the jet to cross-flow momentum flux ratio being maintained at 4.0. Since the momentum ratio at the injection plane is the same for both thermocouple and five hole probe tests it may also be assumed that the momentum of the flow at a given location in the dilution annulus is also the same. This assumption reflects the findings of previous
workers outlined in section (1.2) who indicated that the most important variable determining the trajectory and mixing behaviour is the jet to cross-flow momentum ratio. Although Kamotani and Greber[51] and Srinivasan et. al.[20] indicated a slight independent influence associated with the density ratio, this was only small and can be neglected at the present operating conditions. Thus, if the trajectory and momentum distribution associated with the heated and unheated jets are the same then

\[ \rho_h u_h^2 = \rho_{uh} u_{uh}^2 \]

where \( \rho_h, V_h, \rho_{uh}, \) and \( V_{uh} \) are the local density and velocity values for the heated/unheated case. For ease of understanding though, it is more preferable to present the data in terms of a velocity distribution rather than momentum contours or vectors. However, although the heated jet velocity profiles could be calculated from the isothermal measurements using the formula

\[ u_h = u_{uh} \left( \frac{\rho_{uh}}{\rho_h} \right)^{0.5} \]  

(3.1.4)

so that

\[ u_h \propto u_{uh} \left( \frac{T_h}{T_{uh}} \right)^{0.5} \]

this has not been done, thereby avoiding additional measurements and monitoring of temperatures. Instead, the velocity profiles presented are for unheated jets upon which the measurements have been made and these are derived from the calculated dynamic heads and by assuming a constant density equal to that of the atmospheric value.

3.2 Performance Parameters

The primary objective of this investigation is to assess the consistency of the temperature patterns around the annulus which are produced by the mixing of the heated dilution jets with the relatively cold cross-flow. In order to do this it is necessary to consider the minimum geometric segment which is repeated around the annulus. In a combustion chamber such a segment may include a fuel injector together with several primary, intermediate and dilution holes. However, in this investigation the simplified geometry together with the uniform upstream approach conditions mean that the 16 dilution holes form geometrically similar segments that are repeated every 22.5° around the annulus, with each segment containing a single dilution jet. It is the similarity of the temperature patterns
produced by each segment that must be quantified and assessed. In order to do this the concept of a block distribution is introduced\[^{[33]}\], the physical size of which corresponds to the minimum geometric segment. At any downstream station a block temperature $T_b$ is obtained by averaging the temperatures measured at the same position relative to the dilution hole in each 22.5° segment (Fig.3.3). Thus, each block temperature is associated with a specific $(r, \phi^*)$ location and so by calculating the block values at all the traverse locations a block distribution is obtained which represents the mean of the temperature patterns produced by the segments around the annulus. In addition to calculating a block distribution for the complete annulus, it follows that a mean distribution can be calculated for any number of segments within which detailed measurements have been made. For example, a 135° sector of the annulus contains 6 jets, so that each block value is an average of 6 recorded temperatures and in this case the block distribution is the mean pattern for that sector of the annulus. Thus, at any downstream measurement plane a block temperature is calculated using the formula

$$T_b(r, \phi) = \frac{1}{J} \sum_{j=1}^{J} T(r, \phi)_j$$ \hspace{1cm} (3.2.1)

or in non-dimensional form

$$\theta_b(r, \phi) = \frac{1}{J} \sum_{j=1}^{J} \theta(r, \phi)_j$$

where $J = \text{Number of jets (i.e. segments) of interest}$.

Since each segment is geometrically similar the temperature patterns around the annulus should be virtually the same. In reality this is not the case and the consistency of mixing around the annulus is assessed by comparing the differences in the temperature distribution of each segment from that of the block distribution.

The overall consistency of mixing is quantified by calculating the standard deviation of the temperature patterns around the annulus from that of the mean (i.e. block) distribution. Thus, within each segment and at the same position relative to each dilution hole the temperatures are compared with the relevant block value. This can be repeated at all the traverse locations at which data is recorded and a total standard deviation calculated. For ease of analysis the standard deviation from that of the block distribution is normally calculated for each radius at which data was collected, and so for a sector containing $J$ number of jets (or segments)

$$\text{Standard deviation} (r) = \frac{1}{(T_j - T_c)} \frac{1}{J} \frac{1}{N} \sqrt{\sum_{j=1}^{J} \sum_{i=1}^{N} (T_i - T_{b_j})^2}$$ \hspace{1cm} (3.2.2)
Data Analysis and Performance Parameters

or

\[
\text{Standard deviation } (r) = \frac{1}{\sqrt{N}} \sqrt{\sum_{j=1}^{J} \sum_{i=1}^{N} (\theta_i - \theta_{j, i})^2}
\]

where \( N \) is the number of data points at that radius.

The temperature at any location in the flow field is a function of the amount of cross-flow and dilution air that is present in that small volume of fluid together with the original temperatures of the 2 air streams ie. from section (3.1)

\[
m'T = m'_j T_{i, \text{meas}} + m'_c T_{c, \text{meas}}
\]

so that \( T = f (m'_j, m'_c, T_j, T_c) \)

The use of non-dimensional temperatures means that the standard deviation derived from eqn.(3.2.2) is independent of the cross-flow \( (T_c) \) and dilution air \( (T_j) \) temperatures upstream of the injection plane. However, the temperature at a given location is also a function of the amount of cross-flow and dilution air at that position in the flow field and will therefore vary if the amount of heated air injected is altered relative to the cross-flow. It therefore follows that the standard deviation parameter defined above is also a function of the mass of dilution air injected into the cross-flow. For example, with no air injected through the dilution holes the temperature distribution within each segment is uniform and equal to that of the cross-flow temperature so that the standard deviation is zero. Differences in temperature from that of the block distribution will therefore only occur within each dilution jet where the two air streams are mixing, as indicated by an increase in temperature above that of the 11°C cross-flow value. As the amount of dilution air entering the cross-flow annulus is increased so the dilution jets occupy a larger proportion of the annulus and hence the standard deviation increases. This parameter is therefore only evaluated from temperatures recorded within each dilution jet where mixing has taken place and the boundary of each jet is defined using a non dimensional temperature profile:

\[
\theta' = \frac{T - T_{\min}}{T_{\max} - T_{\min}} = 0.1
\]

where \( T_{\max} \) and \( T_{\min} \) are the maximum and minimum temperatures recorded at the traverse plane being considered and \( T \) represents the boundary of the jet at that location. Since at the measurement planes outlined in this investigation there is always some cross-flow which has not mixed with the dilution air, the minimum temperature corresponds to that of the reference cross-flow value. Thus

\[
\theta' = \frac{T - T_c}{T_{\max} - T_c} = 0.1
\]

33
This definition of the jet boundary has been used by many workers and so the standard deviation is only calculated using temperatures which are above that of the boundary value, which therefore represents regions within which mixing has taken place and where asymmetries will occur. By this means variations in the amount of dilution air are accounted for and comparisons can be made of results obtained from tests with different ratios of cross-flow to injected mass flow. Thus

\[
\text{Standard deviation } \sigma(r) = \frac{1}{N} \sqrt{\sum_{j=1}^{J} \sum_{i=1}^{N} (T_{ij} - \bar{T}_j)^2}
\]

or

\[
\text{Standard deviation } \sigma(r) = \frac{1}{N} \sqrt{\sum_{j=1}^{J} \sum_{i=1}^{N} (\theta_{ij} - \bar{\theta}_j)^2}
\]

where \( N \) is the number of data points within the jet boundary (\( \theta' > 0.1 \)) at that radius and \( J \) is the number of jets being considered.

It is the above definition which is used to assess the mixing performance of the dilution jets around the annulus in this investigation. The traverse plane at which this parameter is evaluated should be quoted with the results since the standard deviation is a function of the distance \((X)\) downstream of the injection plane. Close to the injection point \((X=0.0)\) no mixing will have taken place between the cross-flow and dilution fluid and hence the standard deviation is small. This will increase rapidly and reach a peak value as asymmetries are introduced by the mixing in the near field region close to the injection plane. Eventually as the downstream distance \((X)\) tends to infinity a uniform temperature distribution is obtained and the standard deviation tends to zero. It should be noted that as an alternative the standard deviation could be calculated using non-dimensional temperature profiles, so that the term \((T_j-T_c)\) in eqn.(3.2.5) would be replaced by \((T_{\text{max}}-T_{\text{min}})\). Workers such as Keffer and Baines\(^2\) and Srinivasan et. al\(^20\) have indicated that outside of the near field mixing region, downstream of the jet potential core, there is similarity of the non-dimensionalised profiles. However, the major disadvantage of such a definition is that the standard deviation is based on the relative temperatures at that specific plane being considered. For example, as the dilution air mixes with the cross-flow a 0.5°C variation in temperature from that of the block value would result in a large standard deviation being calculated if the maximum temperature \((T_{\text{max}})\) is only 1°C above that of the minimum \((T_{\text{min}})\) value. For practical applications it is the variation in temperature around the annulus relative to that of the fluid temperatures upstream.
Data Analysis and Performance Parameters

of the injection plane \((T_j - T_c)\) that is of importance. The parameter used in this investigation (eqn.(3.2.5)) is therefore of greatest significance to the combustion engineer who, for a given set of inlet conditions needs to consider the overall system length and the required asymmetries that can be tolerated by the downstream turbine stage.

The symmetry of a temperature pattern produced by an individual dilution jet can also be quantified using the standard deviation concept since the amount of distortion is reflected by differences in the temperature distribution either side of the hole centre-plane. At any radius, the temperature recorded at a given \((\phi^*)\) can be compared with the corresponding \((-\phi^*)\) value on the other side of the jet (Fig.3.4), and these differences can be summed to give the jet distortion:

\[
\sigma_j = \frac{1}{N} \sum_{i=1}^{N} \left( T_i - T_{\phi} \right)^2
\]

where \(N\) is the number of data points \((\theta^* > 0.1)\). This parameter can be quantified at specified radii and the radial variation of the distortion analysed \((\sigma_j (r))\) or alternatively the total distortion of the jet can be calculated \((\sigma)\). At any traverse plane the distortions of the various temperature patterns around the annulus can therefore be compared and the effects of the changes in hole geometry on individual temperature patterns can also be assessed by this means.

In addition to assessing the consistency of the temperature patterns around the annulus, the degree of mixing of the jets as they pass down the dilution annulus can be described by several temperature distribution factors (TDF’s), and again these parameters can be calculated for the complete annulus or a sector containing several jets.

Maximum TDF \((r)\) = \(\frac{T_{\text{max},r} - T_c}{T_j - T_c} = \theta_{\text{max},r}\) (3.2.7)

where \(T_{\text{max},r}\) is the maximum temperature at a given radius

Mean TDF \((r)\) = \(\frac{T_{r} - T_c}{T_j - T_c} = \bar{\theta}_r\) (3.2.8)

where \(T_{r}\) is the area weighted mean temperature at a given radius
Data Analysis and Performance Parameters

Block Maximum TDF ($r$) = \( \frac{T_{\text{max}_b} - T_c}{T_{j} - T_c} = \theta_{\text{max}_b} \)  \hspace{1cm} (3.2.9)

where $T_{\text{max}_b}$ is the maximum block temperature at a given radius.

The difference between the block maximum and mean temperature distributions indicates the degree of mixing between the cross-flow and dilution air, so that as the distance downstream of the injection plane tends to infinity so the difference between these profiles tends to zero. Since the maximum measured and block temperatures at a given radius can occur at any circumferential location it should be noted that the maximum TDF parameters do not represent physical profiles.

As already outlined, deviations in temperature within a given segment from that of the block distribution reflect circumferential irregularities in the temperature pattern around the annulus. In tests where only a limited amount of measurements are available due to a relatively coarse grid of data points, the standard deviation parameter cannot be used to give an accurate assessment of the temperature irregularities. As an alternative, the temperature asymmetry factor (TAF) can be used to evaluate the irregularities which is based on the difference between the highest temperature recorded at a given radius and the maximum block value at that same radius.

\[ \text{TAF} (r) = \frac{T_{\text{max}_r} - T_{\text{max}_b}}{T_{j} - T_c} = \theta_{\text{max}_r} - \theta_{\text{max}_b} \]  \hspace{1cm} (3.2.10)

This parameter reflects the 'hot-spots' relative to the normal variations expected from the mixing of the dilution jets around the annulus.

3.3 Vorticity definitions

The flow field downstream of a transverse jet issuing into a cross-flow is dominated by several counter-rotating regions of streamwise vorticity and several workers have therefore used the concept of vorticity to explain the structure and behaviour of such a jet. In this investigation mean vorticity values are calculated from velocity data derived from 5 hole probe measurements and are used to assist in understanding the vortex dominated flow field recorded downstream of the injection plane and its effect on jet mixing. Furthermore, the concept of vorticity can be used to explain differences between the structure of the multiple jet configuration considered here and that of the well documented single jet issuing into an unconfined cross-flow.

Due to the large injection wall radius, the vorticity downstream of a dilution jet is
calculated using formulae based on a Cartesian coordinate system so that

Streamwise vorticity \( \Omega_x = \frac{\partial W}{\partial Y} - \frac{\partial V}{\partial Z} \) \hspace{1cm} (3.3.1(a))

Radial vorticity \( \Omega_y = \frac{\partial U}{\partial Z} - \frac{\partial W}{\partial X} \) \hspace{1cm} (3.3.1(b))

where each gradient is evaluated from the mean velocity data to which is fitted a 3 term Lagrange polynomial\(^{[48]}\) in the region of interest.

The equations for the rate of change of vorticity are derived from the Navier-Stokes equations (Appendix 1) and so for the \( x \) component:

\[
\frac{\partial \Omega_x}{\partial t} + U \frac{\partial \Omega_x}{\partial X} + V \frac{\partial \Omega_x}{\partial Y} + W \frac{\partial \Omega_x}{\partial Z} = \Omega_x \frac{\partial U}{\partial X} + \Omega_y \frac{\partial U}{\partial Y} + \Omega_z \frac{\partial U}{\partial Z} + u \left( \frac{\partial^2 \Omega_x}{\partial X^2} + \frac{\partial^2 \Omega_x}{\partial Y^2} + \frac{\partial^2 \Omega_x}{\partial Z^2} \right)
\]

This can be written in substantial derivative form as

\[
\frac{D \Omega_x}{Dt} = \Omega_x \frac{\partial U}{\partial X} + \Omega_y \frac{\partial U}{\partial Y} + \Omega_z \frac{\partial U}{\partial Z} + u \left( \frac{\partial^2 \Omega_x}{\partial X^2} + \frac{\partial^2 \Omega_x}{\partial Y^2} + \frac{\partial^2 \Omega_x}{\partial Z^2} \right) \hspace{1cm} (3.3.2)
\]

The substantial derivative term on the left side of equation (3.3.2) is the rate of change of vorticity for a particle of fluid as it moves along a streamline in the dilution annulus. The first terms on the right of equation (3.3.2) represents the production of vorticity due to the stretching of the vortex lines, whilst the second term indicates the diffusion of vorticity due to viscous effects.

A greater explanation of the above equations and how they are derived is given in Appendix 1.

3.4 Presentation of data

Conventional 2-D graphs are used to summarise the results obtained from thermocouples located in the dilution annulus, whilst contour plots illustrate the development of temperature patterns downstream of the injection plane. The temperature contours are plotted in terms of the non-dimensional temperature \((\theta)\) and are derived from a linear interpolation of the experimental measurements.
Data Analysis and Performance Parameters

All the velocity data derived from the isothermal tests are non-dimensionalised with respect to the jet velocity \( V_j \) measured across the exit plane of the dilution holes. Contours of constant velocity are used to indicate the magnitude of the velocity components perpendicular to the traverse plane and, like the temperature contours, are derived from a linear interpolation of the data. Resolved components of velocity in the traverse plane are presented in the form of vectors, where the flow direction and magnitude of velocity at points in the plane are indicated by the length and direction of the arrows. Streamlines should not, however, be inferred from the velocity vectors since the flow field is strongly 3-dimensional, particularly close to the injection plane. Despite the use of several probes with different geometries which could also be yawed into the flow direction, there are regions of the flow field which remain outside the probe calibration ranges of \( \pm 35^\circ \) in pitch and yaw. Data points are omitted in these ranges although in certain areas where only one flow angle is outside the calibration limits an indication of the flow direction in one plane may be given, and where this has been done the data is represented by a dotted line. Contours of constant vorticity can also be deduced from the velocity data and these are non-dimensionalised using the jet velocity \( V_j \) and the dilution hole diameter (D). No attempt is made to interpolate these contours in regions of the flow field where velocity measurements are limited due to the finite probe calibration ranges.

All the diagrams are derived from measurements made in cylindrical \((r, \phi)\) coordinates which have to be presented in 2 dimensional form. For data collected in traverse planes perpendicular to the centre and outer casings a true 2 dimensional view of the flow field can be presented, but this is not the case for measurements made in planes parallel to the casing at a fixed radial location. The surface formed by the experimental grid of data points (Fig.3.5) is therefore 'flattened' out for presentation purposes and drawn as a 2 dimensional surface. No correction is made to account for this procedure since it is assumed that the distortion of the data being presented is small due to the relatively large radii at which the measurements are made.

3.5 Estimation of experimental errors

For the measurement system used it is necessary to assess the accuracy of the system and establish the resolution limits. It is assumed in this investigation that the correction procedures outlined in section (3.1) are accurate and that the experimental errors associated with the measurement system are related to the instrumentation and rig hardware. Since accuracy is a function of the flow field being measured, the experimental errors are calculated for the \( X/D=2.0 \) traverse plane where the mixing performance of the dilution jets is mostly evaluated.
3.5.1 Temperature measurements

The mixing performance is assessed from thermocouple measurements of the temperature field in the dilution annulus and so the accuracy of these measurements is of fundamental importance to this investigation. It is assumed that the finite size of the thermocouple intruding into the dilution annulus has negligible effect on the temperature field being measured, but it has already been mentioned that the values measured are a function of velocity as well as static temperature.

\[ T = t + \frac{u^2}{2C_p}Rf \]

\( Rf \) - recovery factor of the thermocouple

For the bare thermocouple type sensors used in this investigation the recovery factor has a minimum value of approximately 0.6\(^{[46]}\). A maximum velocity of approximately 30m/s is recorded across the exit plane of the dilution holes which would result in a component of approximately 0.3°C being measured above that of the static temperature. Most measurements however are conducted in the cross-flow annulus at speeds of up to approximately 23m/s leading to variations due to velocity effects of less than 0.2°C. Although this represents an error in the static temperature field being measured, it has negligible effect in terms of the mixing performance parameters and the objectives of this investigation. The net effect is that differences in the temperature patterns around the annulus will also have a relatively small component due to variations in the velocity field associated with each jet.

The thermocouple signals are connected to Comark digital thermometers which have a resolution of 0.1°C, and the output signal from this equipment is supplied to a 12 bit ADC which therefore has a quantising error of 0.05°C for a 100°C measurement range. These effects result in differences of up to 0.15°C in the temperature patterns around the annulus which are attributable to experimental error. However, further errors can also be identified the magnitudes of which cannot be easily estimated. These are associated with the accuracy of the thermocouple positioning in the flow field and the ability to maintain the correct momentum flux ratio during a test. This can lead to significant errors, the magnitude of which varies with the type of test being conducted.

a) Complete annular surveys -

For each set of readings the 16 thermocouples are located at the same position relative to each dilution hole and any recorded asymmetry is therefore logged at virtually the same flow conditions. The major source of error is therefore the accuracy with which the
individual thermocouples can be located in the dilution annulus and this has been estimated to be in the order of 1mm (0.039D). Around the periphery of each jet where temperature gradients are high, this will have a significant effect on the temperatures values recorded. For example, downstream of the injection plane where the mixing performance is evaluated, the misalignment of a thermocouple by 1mm can result in a difference in temperature of 2°C from that of the other values recorded. Furthermore, the different characteristics of each thermocouple may introduce an additional error of 0.15°C which, together with the aforementioned resolution and quantising errors, gives a total variation in the temperature measurements of 2.3°C. This represents 5.2% of the difference between the jet reference temperature of 55°C and the cross-flow reference value of 11°C.

b) Sector surveys:-

Since the same thermocouple is used to survey the temperature pattern associated with several dilution jets there are no errors due to differences in thermocouple characteristics and the traverse mechanism used for these tests also gives high positional accuracy. It is estimated that a probe can be located to within 0.1mm of its desired position in the flow field which in regions of high temperature gradients represents an error of 0.2°C. Since each jet is traversed individually there is some fluctuation in the temperatures measured due to the finite limits of ±0.04 within which the momentum flux ratio can be maintained relative to its nominal value of 4. The order of magnitude of the error due to this effect can be assessed by using empirically based formulae to calculate the change in jet penetration associated with the variation in the momentum ratio. Using the definition of Srinivasan[20] which defines the centreline of a heated jet as that of the loci of maximum temperature:

\[
\frac{Y}{D_j} = 0.539 \left( \frac{S}{D_j} \right)^{0.25} \left( \frac{H}{D_j} \right)^{0.14} \left( \frac{X}{D_j} \right)^{0.38} \left( \frac{X}{D_j} \right)^{0.17} e^{-b}
\]

where

\[
b = 0.091 \left( \frac{X}{H} \right)^2 \left( \frac{H}{S} - \frac{\sqrt{3.5}}{3.5} \right)
\]

and

\[
D_j = \sqrt{C_D} D
\]

At the downstream plane of interest the jet penetration is 1.445D and this will change by up to ±0.007D due to experimental variations of the momentum flux ratio. Using a similar argument to those already outlined this represents a variation in temperature of 0.35°C in regions of high temperature gradients. Together with the other components already identified
the total temperature variation that can be attributed to experimental error is 0.65°C which is 1.5% of the difference between the jet and cross-flow reference conditions.

The above estimates indicate that even with exactly the same temperature patterns around the dilution annulus the mixing performance parameters can indicate variations of up to 5.2% or 1.5% (depending on the type of test being conducted) due to differences in the measured temperatures caused by experimental errors. However, the magnitude of the measurement errors is a function of the temperature gradient and therefore varies depending on the location of the thermocouple in the dilution annulus. The above estimates are for regions of high gradients and therefore represent the maximum temperature variations that can occur. Since the mixing performance parameters are based on a number of measurements within the dilution jets the errors of 5.2% or 1.5% represent the absolute maximum contribution that can be made to the performance parameters due to experimental errors.

3.5.2 Velocity measurements

Miniature 5 hole pressure probes are used to provide information on the velocity flow field in the dilution annulus, indicating how the unheated dilution jets mix and the flow features which dominate jet behaviour. However, since the mixing performance is assessed in terms of temperature no detailed investigation of the 5 hole probe performance limits have been undertaken, since this could be the basis of a separate and lengthy research program. Instead, the work of other people has been used to assess likely probe accuracy and, where possible, errors have been minimised by adoption of a suitable experimental technique.

The 5 hole probes used in this investigation are calibrated for a range of flow angles up to ±36° in both pitch and yaw using a purpose built facility which inclines a probe in an air jet of known conditions. To check the accuracy of the calibration technique tests were performed by Wray in this facility and indicated negligible errors in terms of the angles and velocities measured. When used for measuring flow fields in the dilution annulus however, errors are introduced when traversing close to the casings. Wall proximity effects have been investigated by Tamigniaux and Oates who found that with probes of conical geometry a maximum error of 2° can occur, whilst Sitaram et. al expressed the error in terms of a 1% variation in the velocity magnitude. These errors however are restricted to within one probe diameter of the casing and so for the vast majority of measurements the errors due to wall proximity effects are negligible. The effect of turbulence on 5 hole probe measurements has also been assessed by Sitaram and indicated likely errors in velocity of only 0.33% for turbulence intensities of 10%. The error in velocity though will be larger than that found by Sitaram since Andreopoulos and Rodi have indicated turbulence intensities greater than 30% close to the injection plane, although this occurs in the near field wake region where measurements are limited anyway by the local flow angles.
The physical size of the probe has 2 effects on the accuracy of the measurements close to the injection plane. Firstly, the probe represents an obstruction to the fluid and so the measured flow field is altered by the presence of the probe. Although as small as possible, the size of the probe is limited by the bore of each pressure tube and any further reduction from the present size leads to blockages of the tubes due to ingestion of dust particles. The 1.71mm outside diameter of the probe represents 2.2% of the dilution annulus height and corresponds to 0.5% of the dilution hole area. Of greater significance can be the errors that are introduced when measurements are made in a highly sheared flow due to the finite distance between the measurement holes on the probe head. Although Sitaram cites this as the more likely dominant error for 5 hole probes the problem is minimised in this investigation by interpolating the individual probe hole readings in both the radial and circumferential directions using a cubic spline technique, as outlined in section (3.1). The errors due to the finite size of the probe are further minimised by making the distance between the data points as small as is practically possible, particularly in regions where flow features are small and pressure gradients high. In such regions the distance between the data points is of the order of the diameter of the probe tip.

The pressures sensed by the 5 hole probes are converted into voltage readings by transducers and the most significant error introduced by this process is the transducer drifts. After each test the operational drift of each transducer is recorded and found to be within ±1mV of its starting value. The magnitude of the error that is introduced is a function of the velocity being measured, and for a given flow rate the magnitude of the error introduced can be estimated by changing a raw data file to simulate such effects. For example, using transducers with a range of ±100 mmH₂O to measure flow moving at 30 m/s (=V₁) an error in velocity of approximately 0.2% is introduced, whilst at a lower velocity of 15 m/s the error is approximately 0.7%. At the higher velocity, changes in flow angles of approximately 0.05° occur, which increase to 0.2° at the lower velocity of 15m/s.

It is difficult to estimate the magnitude of the errors associated with the measured velocity field and it is acknowledged in certain regions these could be significant. However, in the complex flow fields that are present close to the injection plane the errors are large even with alternative types of instrumentation such as a triple hot wire anemometry system. It is only a 3 spot L.D.A. system which could possibly produce accurate measurements and the expensive nature of this equipment precludes its use from this investigation. There are a number of methods though of checking the qualitative results obtained from the 5 hole probes. Repeating the test results eliminates errors due to transducer drifts and increases confidence in the average measurements of time dependent flow features such as vortices. Flow visualisation techniques such as wool tufts, vortex detectors and the injection of oil vapour helps to confirm the presence of vortices and establish recirculation regions, separation points etc. Finally, the good correlation between the temperature distribution and
the flow features revealed by the 5 hole probes indicate that even though the magnitude of the velocity measurements may be subject to error, the qualitative description of the flow field is correct.
Chapter 4: Results and Discussion

4.1 Rig calibration tests 45

4.2 Temperature measurements (X/D=2.0) 46

4.3 Flow at exit of the dilution holes (Y/D=0.05) 48

4.4 Downstream velocity field (X/D=2.0) 50

4.5 Initial development of a jet 51
  4.5.1 Initial jet development - Hole 8 51
  4.5.2 Initial jet development - Hole 10 53

4.6 General flow characteristics of multiple jets in
   a confined cross-flow 54
   4.6.1 Bound vortex system 54
   4.6.2 Cross-flow vortex system 55

4.7 Major factors influencing the dilution hole exit velocity profile 56

4.8 Modified dilution hole designs 60
   4.8.1 3 hole sector tests 61
   4.8.2 8 hole sector tests 63
   4.8.3 Hole discharge coefficients 64
4.1 Rig Calibration Tests

The uniformity of the flow approaching each dilution hole is assessed from pitot probes and static pressure taps located in both the cross-flow and feed annuli.

In assessing the uniformity of the velocity profiles around the cross-flow annulus no dilution air is supplied to the feed annulus, and in this way the performance of the intake and cross-flow passage is established in the absence of any jet/cross-flow interactions. At a distance of 6 hole diameters upstream of the injection plane the radial velocity profiles measured at 3 equi-spaced circumferential locations are in excellent agreement and indicate the nature of the flow field approaching the dilution holes (Fig. 4.1.1). At the same upstream location, circumferential velocity profiles measured at radii corresponding to 33%, 50% and 66% of the cross-flow annulus height indicate that at these locations a maximum variation in velocity of only 1.5% is present around the annulus. These locations correspond to the inner, centre and outer edges of the dilution jets at 2 hole diameters downstream of the injection plane and is where Wilson[42] recorded large temperature irregularities around the dilution annulus.

Information on the uniformity of the flow around the dilution hole feed annulus is restricted due to the limited amount of access that is available to this passage. A rake of 5 pitot probes and a static pressure tap provide the means of obtaining velocity profiles 3.5 hole diameters upstream of the injection plane, and all measurements are performed with no heat being supplied to the dilution air. At a given radial location circumferential variations in velocity of up to 5% occur around the annulus with no cross-flow present and the radial velocity profiles at 3 equi-spaced circumferential locations show the more fully developed nature of the flow in the feed annulus (Fig. 4.1.2). The increased influence of the boundary layers on the flow field is probably responsible for the larger velocity variations in this passage when compared with the cross-flow annulus. These measurements however still indicate a highly uniform flow field approaching the dilution holes which is much better than that found in the combustor tested by Shaw[35]. Furthermore, there is no correlation between the location of the maximum velocities in the feed annulus and the temperature irregularities that have been measured in the dilution annulus.

With both air-streams present an increase in the circumferential variation of the cross-flow velocity is to be expected since the localised blockage created by each jet is sensed by the upstream flow. However, the circumferential velocity variation in the feed annulus also increases from 5% to 9% and illustrates how the cross-flow can effect the flow field in the feed annulus passage.
4.2 Temperature measurements (X/D = 2.0)

Having established the quality of the flow approaching the dilution holes the overall irregularity of the downstream temperature patterns can be assessed from 16 thermocouples located around the mixing annulus. Initial results however were somewhat surprising since, despite the uniform approach conditions, relatively large temperature irregularities were recorded around the annulus. At a distance of 2 hole diameters downstream of the injection plane a maximum temperature asymmetry factor (TAF) of 13.9% was recorded at a non-dimensional radius ($r_n$) of 0.67. At the reference operating conditions this represents an increase in temperature of 6°C above that of the block maximum value of approximately 20°C, and the radial location indicates that the irregularities are associated with the outer edges of the jets. Rather than collect data over the complete annulus to indicate the repeatability of this result, measurements were conducted only at a non-dimensional radius of 0.67 from which a block distribution can be derived for that radial location and an asymmetry factor calculated. The values of 15.0% and 14.2% indicate the repeatability of the test results at the maximum asymmetry radius for this initial series of tests.

Upon dismantling the facility following these tests however an inspection of the instrumentation section revealed significant variations in the location of several thermocouples with respect to their associated dilution holes. By resetting each thermocouple to its datum position and repeating the temperature measurements around the annulus a maximum asymmetry factor of 8.3% was recorded at a non-dimensional radius of 0.33. This lower value reflects the reduced positional errors from that of the last test series and illustrates the problem with this particular test method. The required setting of each thermocouple to its datum position is critical and, together with the lack of sophisticated traverse mechanisms for locating the 16 thermocouples during a test, can lead to large variations in the mixing performance parameters. The change in the maximum asymmetry value between the 2 series of tests is of the same order of magnitude as that estimated in section (3.5) which considered the effects of positional accuracy on the mixing performance parameters.

The temperature distribution and asymmetry factors calculated from the 2nd series of test data indicate the degree of mixing and the overall mixing performance at this traverse plane (Table 4.2.1). The radial variation and magnitude of the block maximum distribution factor (Fig.4.2.1) indicates a significant amount of mixing has already taken place, with the values of this parameter being well below that of the non-dimensional jet temperature ($\theta$) of 1.0 at the injection plane. The difference between the block maximum and mean profiles shows the expected variation in temperature around the annulus due to the temperature patterns associated with the mixing of the relatively hot jets. The difference between the block maximum TDF and the maximum measured temperatures illustrates the significant local
temperature maxima that occur and which are associated with specific jets whose peak temperatures are significantly greater than the average maximum values around the annulus. This difference represents the temperature asymmetry factor, the magnitude of which is significant when compared with the block maximum and mean temperature values. It is such local peaks in temperature above that associated with the normal temperature variations around the annulus which have serious consequences for the nozzle guide vanes in a gas turbine engine.

Having quantified the temperature irregularities around the annulus the contours of constant non-dimensional temperature (Fig.4.2.2) illustrate the characteristic kidney shaped cross-section of the dilution jets and also differences in the individual temperature patterns. The largest temperature asymmetries are associated with the core regions where it can be seen that the temperature contours of certain jets exhibit an apparent rotation or 'twist'. This indicates the displacement of relatively hot core fluid to a lower radius than would otherwise be the case and changes the temperature distribution of the jet core. The effect is illustrated by comparing the temperature pattern of the jet issuing from hole 10 with that of the block distribution for the annulus. Since the core of each jet can rotate in either direction and by differing amounts, so this leads to variations in the temperature patterns from one jet to another.

Comparison of the present results with that of Wilson's data[42J at X/D=2.0 (Fig.4.2.3) indicates the similarity of the temperature patterns measured in each test series. Thus, despite changes in rig geometry which results in different conditions in the feed annulus, the same apparent distortions are observed in the temperature contours which reflect variations of the temperature patterns around the annulus. Furthermore, Wilson also tested at X/D=3.0 and X/D=4.0 (Fig.4.2.4 and Fig.4.2.5) and these results show how the distortion and displacement of each jet core at X/D=2.0 continues to be observed further downstream. Hence, similar levels of distortion in the overall temperature distribution continue downstream as the 2 air streams mix. This confirms the findings of Moussa et. al.[111] who concluded that the development of a jet in the near field region (X/D<1.0) characterises the subsequent behaviour of that jet further downstream. Rather than make a large number of measurements at many locations along the dilution annulus, this investigation therefore concentrates on the temperature distribution close to the injection plane and assesses the mixing performance at the traverse station X/D=2.0. Hence, the temperature patterns measured at this location are assumed to reflect the distortion that will continue to occur along the mixing annulus.

The variations in the temperature patterns recorded cannot be attributed to experimental errors and the distributions indicate their are significant differences in the way individual dilution jets mix. However, since data is collected in radial and circumferential increments of 0.25D and 2.5° respectively there is a limit as to what flow features can be resolved. The
Results and Discussion

temperature patterns that have been presented are derived from relatively large interpolations of the experimental data which is responsible for the 'jagged' appearance of the contours. In order to obtain more detailed data on the mixing characteristics of the jets a larger number of closely spaced temperature measurements is therefore required. Hence, sector surveys were carried out in a segment of the annulus containing holes 7 to 12 which include jets which exhibit the largest distortions and temperature differences relative to the block distribution.

The results obtained using the finer experimental grid of data points (Fig.4.2.6) clearly show that, despite the uniform flow conditions approaching the injection plane, there are significant differences in the temperature patterns produced by the dilution jets with the largest distortions being produced by the flow issuing from holes 8 and 10. Some of these results have already been presented by Carrotte and Stevens[52], and they clearly illustrate the different amounts of rotation or apparent 'twisting' of the kidney shaped cross-section of each jet which is a major factor in producing the different temperature patterns around the annulus. The smoothness of the non-dimensional temperature (θ) contours derived from this test method reflects the quality of the measurements and the ability to resolve relatively small flow features. Furthermore, the degree of repeatability of the test data is indicated by the contours obtained from a second series of measurements downstream of holes 9 and 10 (Fig.4.2.7). These results confirm the repeatability of the distortions exhibited by the jets and show that these effects are not sensitive to the method of starting the test rig or of tests being conducted on different days at varying ambient conditions.

4.3 Flow at exit of the Dilution holes (Y/D = 0.05)

One possible explanation for the shape of the temperature contours is that a rotational component is present in certain jets as they discharge from the dilution holes. To test this theory traverses with a 5 hole pressure probe were carried out across the exit plane of the dilution holes at Y/D =0.05. Initial tests were carried out with the outer casing removed so that no cross-flow was present, and under these conditions the velocity vectors show that vortex structures are present in the rear half of certain dilution holes (Fig.4.3.1). For example, in the case of hole 9 a well defined vortex with a clockwise sense of rotation is positioned to the left of the hole centreline, whereas a vortex with an opposite sense of rotation is similarly displaced but to the right of the centreline of hole 10. The strength of these vortices can be assessed from the velocity vectors and the radial contours indicating the magnitude of the velocity components normal to the traverse plane. For hole 10, rotational velocity components of up to approximately 16m/s are recorded in the vortex which, in comparison with the radial velocity of 18m/s at the centre, indicates the strength of the vortex issuing from the hole. However, pressure measurements within a vortex may be time
dependent since velocity fluctuations can occur due to the nature of the flow field. Despite only mean flow field measurements being possible though, the ability of the 5 hole probe to repeat a test result (Fig.4.3.2) indicates the accuracy with which the exit plane flow features can be measured. Tests indicate that the pattern of vortices in adjacent holes rotating in opposing directions continues around the annulus until interrupted by the pattern depicted in hole 8. Here the flow at the rear of the hole is symmetrical about the centreline with a component in an opposite sense to the direction of flow feeding the hole, either side of which are located two smaller vortices. It will be observed that due to the way in which air is fed to the holes along an approach annulus, even in the absence of a cross-flow, the jets exhibit a pronounced pitch component.

The velocity distributions across the exit planes of the holes are effected by the introduction of the cross-flow (Fig.4.3.3). The well defined vortices though in holes 9 and 10 are still present, but in the rear half of holes 7 and 8 asymmetry has been produced with vortices of anti-clockwise rotation being sited to the left of the hole centralines. A summary of these effects for holes 8 and 10 is given in terms of the variation in pitch angle ($\alpha$) both along the axial centraline ($\varphi=0^\circ$) and the mean value over the face of the jet at a given (X) location (Fig.4.3.4). Deflection of the jet by the cross-flow is indicated by the increase in pitch angle, which is particularly evident in the front half of holes 8 and 10 where the angle increases by about $12^\circ$ and $10^\circ$ respectively. Whereas the values along the centraline in the rear section of the hole are determined essentially by the position of the vortex, the mean values do indicate a reduction in pitch angle with the flow leaving almost normal to the surface. The radial velocity contours for holes 8 and 10 show how the flow in the jet is distorted by the cross-flow even at a distance of $Y/D=0.05$. For both holes the front half of the jet has a decreased velocity and the flow in the rear half is forced to accelerate to accommodate the extra mass flow which is in agreement with the findings of Crabb et. al[12]. Furthermore, in the rear half of the jet the cross-flow causes a widening of the high velocity region and hence the lateral stretching of the jet commences as it is formed.

The flow features that have been observed in measurements across the exit plane of each dilution hole indicate the importance of having representative feed conditions. It is necessary to simulate the annulus formed by the combustor liner and outer casing, so that the approach flow direction is approximately perpendicular to the axial centraline of each hole. The subsequent deflection of the flow as it passes from the feed annulus through each dilution hole must have a major influence on the velocity profile across the exit plane, producing an extremely complex flow field in the ensuing jet which varies in a random manner from one jet to another. Any investigation concerning irregularities in the temperature patterns around a dilution annulus must therefore supply air to the dilution holes from a representative feed annulus, since such flow features are not observed when a row of jets are supplied from a plenum (see section 4.7).
4.4 Downstream Velocity Flow Field (X/D = 2.0)

Having established that vortices are formed in the jets as they issue from the dilution holes, the next phase of the test program was aimed at determining to what extent these vortices influenced the downstream structure of the flow. To this end traverses were carried out normal to the injection wall at a distance 2 hole diameters downstream of holes 7 to 12. One of the difficulties in using such a probe is that large flow angles may be encountered, particularly in the wake region where flow recirculations are present, and so due to the finite range of the 5 hole probes only a limited amount of data is therefore available.

For closely spaced multiple jets issuing into a confined cross-flow the velocity vector plots 2 diameters downstream of holes 7 to 12 indicate the complex nature of the flow field (Fig.4.4.1), and for 2 of the holes tested both the temperature and velocity distributions are also presented for purposes of comparison (Fig.4.4.2). This data indicates that there are significant differences in the flow field measured downstream of the dilution jets in comparison with that of the well documented structure of a single jet issuing into an unconfined cross-flow. Workers such as Andreopoulos and Rodi[14], Moussa et. al.[11] and Fearn and Weston[16] as well as the numerical solutions of Sykes et al[17] have shown that bound vortices are located in the shear layer at the lobes of the jet. In addition, the velocity vectors indicate a clearly defined structure with the flow circulating around these vortex centres (Fig.4.4.3(a)). However, although in the present data there is some evidence of bound vortices being attached to the lobes of the kidney shaped cross-section, these are relatively small when compared with the double vortex structure developing in the wake of each jet. Although the data is limited in this region, the flow appears to roll up underneath each jet resulting in vortex centres being located in the wake (Fig.4.4.3(b)). This results in a highly complex flow field downstream of the injection plane, and these observations are complicated by the distortion or 'twisting' of the jet that occurs about the hole centre-plane. Although in regions of high temperature there is no evidence of the vortices which issue through the dilution holes, there is asymmetry of the aforementioned vortex structures in the wake and at the lobes of each jet. Furthermore, with the possible exception of hole 12 a correlation exists between the distortion or 'twisting' of the temperature contours and the vortex structures, since the side of the jet with the highest concentration of jet (high temperature) fluid corresponds to the location of the vortices of greatest strength. The influence of an asymmetric vortex flow field downstream of the injection plane was also indicated by Srinivasan et al.[22], where vortices of unequal strength were generated by 45° angled dilution slots and were thought to be responsible for producing an apparent rotation of the jet temperature contours.
Since the behaviour of the jet is controlled mainly by pressure forces which cause the bending of the jet and the formation of the double vortex structure, the subsequent shape of the jet must be dependent on the initial momentum distribution of the fluid issuing from the dilution hole. It is therefore reasonable to assume that the non-uniformities measured across the exit planes of the dilution holes could, under the action of the pressure forces, lead to a non-uniform distribution of the jet fluid about the centre-plane of the hole and be instrumental in the formation of vortices of unequal strength. The physical processes which are responsible for the distortion of the temperature contours associated with a jet are clearly complex and may well be influenced by several factors. This is illustrated by the flow issuing from holes 8 and 10 which have very different exit plane velocity distributions but the resulting jets which develop exhibit similar large distortions of their temperature contours. It is also interesting to note that whereas the axial velocity distribution for hole 10 indicates that the core of the jet has 'twisted', in the case of hole 8 the distribution of jet fluid is more symmetric and the velocity contours do not reflect the distortion of the temperature contours. It is therefore quite clear that the mechanisms which cause irregularities in the temperature patterns around the annulus are complicated and are not observed in the well documented single jet, with a plenum feed, exhausting into an unconfined cross-flow.

4.5 Initial development of a jet

Detailed measurements were made on the jets issuing from holes 8 and 10 which, as already outlined, exhibit similar degrees of distortion in their temperature distributions but which have different velocity profiles across their exit planes. Some of these results have already been presented by Carrotte and Stevens[58], the objective being to establish the influence of the exit plane velocity profile on the jet development and how this can lead to irregularities in the downstream temperature distributions.

4.5.1 Initial jet development - Hole 8

Measurements conducted in planes perpendicular to the downstream (X) direction indicate the initial development of the jet as it issues into the cross-flow. Similar to the velocity measurements made by Moussa et. al.[11] on a single jet, the data collected at X/D=0.30 (Fig.4.5.1) illustrates the formation of bound vortices at the lateral edges of the jet and their influence on the temperature distribution in this region. The velocity data at X/D=0.54 (Fig.4.5.2) shows the continued development of these vortices as the jet progresses downstream, being located at the lobes of the characteristic kidney shaped temperature contours which are already visible at this location. However, even at this early
Results and Discussion

stage of jet development the distortion of the temperature distribution is apparent both in the hot core region of the jet and close to the injection wall.

Measurements conducted parallel to the injection wall indicate the influence of the velocity profile, across the exit plane of the dilution hole, on the subsequent development of the jet and its downstream temperature distribution. The development of the complex flow field associated with the feed conditions to the dilution holes is evident in the velocity vector plot at Y/D=0.2 (Fig.4.5.3), which also illustrates the internal movement of fluid within the jet. Fluid in the front half of the jet which has been deflected by the cross-flow is forced to move around the relatively undeflected flow in the rear of the jet, this curvature being indicated by an increase in the downstream velocity component (U). However, due to the behaviour of the flow issuing through the rear of the dilution hole this deflection is not symmetric about the hole centre-plane, as illustrated by the velocity (U) component profiles at this location (Fig.4.5.4). As expected, the increased curvature of the slower moving fluid in the turbulent shear layer at the lateral edges of the jet results in an increase in the downstream velocity component, but this deflection differs from one side of the jet to the other. Fluid issuing through the left (φ<0) side of the hole is deflected much more rapidly than fluid on the opposite (φ>0) side of the centre-plane which has to pass over and yaw around the greater effective blockage associated with the complex flow field issuing through the hole. Immediately downstream of the injection plane at X/D=0.54 the velocity profiles at φ=±2.0° (±0.25D from the centreline) confirm this observation (Fig.4.5.5), with fluid on the right (φ>0) side of the jet exhibiting greater yawing and penetration into the cross-flow annulus. It is these variations in trajectory, which also affect the jet path length to a given downstream (X) location, that are responsible for producing the distortion of the core of the jet.

The development of the bound vortices at the lateral edges of the jet must also be influenced by the trajectory of the jet fluid, and this is evident in the temperature distribution at X/D=0.30 (Fig.4.5.1). Under normal circumstances the vortices act as a transport mechanism, displacing relatively hot fluid to a lower radius and mixing it with the cross-flow. The additional yaw component imparted to one side of the jet (φ>0) increases the lateral spreading of the hot fluid, and affects the characteristics of the vortex as indicated by both the velocity and temperature data at X/D=0.30. This is further illustrated by the temperature distribution parallel to the injection wall at Y/D=0.2 (Fig.4.5.6) which shows the increased lateral spreading of the hot jet fluid on the right (φ>0) side of the jet. This temperature distribution reflects the bound vortex flow field, since the relatively hot regions originating from the lateral edges of the jet are associated with the downward motion of the jet fluid towards the injection wall due to the action of the vortices. This fluid then moves laterally inwards into the jet wake where it is mixed with the cross-flow and its temperature reduced. Since mixing close to the injection wall is controlled by this bound vortex flow field, the different vortex and hence mixing characteristics either side of the jet are therefore
thought responsible for producing the asymmetry observed in this temperature distribution.

The trajectory of the jet fluid and the different bound vortex mixing characteristics not only directly affect the temperature distribution but also result in different amounts of fluid being supplied from either side of the jet into the wake region. This fluid then rolls up to form 2 counter-rotating vortices in the wake region of each jet and therefore explains the differences in the size of these flow features which have been recorded at X/D=2.0 (Fig.4.5.7). Although the near field mixing region (X/D<1.0) is dominated by the bound vortex system attached to the lobes of the jet, this decays rapidly and the velocity data at X/D=2.0 shows that the dominant downstream vortex system is located in the jet wake. The side of the jet with the stronger vortex at this location entrains more jet fluid so producing a recirculating region which transports hot fluid towards the injection wall, thereby ensuring continued distortion of the temperature distribution as the jet progresses downstream.

4.5.2 Initial jet development - Hole 10

The approach velocity in the feed annulus produces a well defined vortex which issues through the rear of hole 10 and which is still present at Y/D=0.2 (Fig.4.5.8). Due to its strength and location a large blockage is created as fluid in the front of the jet is deflected by the cross-flow and this causes a redistribution of this fluid about the hole centre-plane so that the core of the jet becomes displaced to one side. This is illustrated by the temperature distribution at X/D=0.54 (Fig.4.5.9) and also the velocity distribution which indicates a reverse flow region on the right ($\phi'>0$) side of the jet associated with the hole vortex. The redistribution of fluid is also indicated further downstream at X/D=0.72 (Fig.4.5.10) and shows the effect of the vortex issuing through the hole is to cause most of the jet fluid to be deflected to the left ($\phi'<0$) of the hole centre-plane. This is the main cause of the temperature distortion recorded at the X/D=2.0 traverse plane, with the axial (U) velocity distribution also indicating asymmetry due to the displacement of the jet fluid (Fig.4.5.11). Of secondary importance is the influence of the fluid trajectory on the bound vortex system, with a weak vortex developing on the right side ($\phi'>0$) of the jet. Since the hot dilution fluid is deflected away from this region under the action of the hole vortex, the lack of fluid means this bound vortex only has a minor influence on the temperature distributions close to the injection plane (Fig.4.5.12). On the other side of the jet a normal bound vortex develops which transports fluid to a lower radius, this feature decaying rapidly as indicated by the velocity distribution at the X/D=0.72 traverse plane.

In summarising, for hole 8 the variation in fluid trajectory due to the complex flow field issuing through the rear of the hole causes a distortion of the temperature pattern which is enhanced by the downstream asymmetric vortex flow field created by the jet. In the case of hole 10, the blockage created by the well defined vortex issuing through the hole is sufficient
Results and Discussion

to cause a major redistribution of the jet fluid about the centre-plane, with the downstream vortex flow field only having a minor influence on the temperature distortion.

4.6 General flow characteristics of Multiple jets in a Confined Cross-flow

As already indicated there appears to be significant variations between the well documented structure of a single jet issuing into a relatively unconfined cross-flow and the data presented here for a multiple jet configuration. Although the initial deflection process is similar in each case and the characteristic kidney shaped cross-sections are also observed, it is the downstream flow field where differences occur. A diagram of the flow characteristics observed downstream of a row of jets injected into a confined cross-flow is presented (Fig.4.6.1), and this has been derived from measurements made on the fluid issuing from holes 8 and 10. Although the near field region is dominated by the bound vortex system, this decays rapidly so that the downstream flow field (X/D>1.0) is dominated by a vortex system situated in the jet wake, and for purposes of identification this is referred to as the cross-flow vortex system. As already outlined this is somewhat different to the case of a single jet in an unconfined cross-flow where it is assumed the downstream flow field is dominated by a bound vortex system attached to the lobes of the jet. Some of these differences can be explained by considering the vorticity characteristics of the flow and how they are affected by the different velocity field surrounding the row of jets in the confined cross-flow.

4.6.1 Bound vortex system

At the injection plane vorticity is generated by the high radial velocity gradients (∂V/∂Z and ∂V/∂X) around the periphery of the jet which are convected into the stream wise direction. In the multiple jet configuration the blockage caused by the row of jets in the dilution annulus leads to an acceleration of the cross-flow between each jet. Thus streamwise vorticity (Ωₓ) is intensified by the extensional strain rate (∂U/∂X>0) producing the well-defined vortices that have been seen in the velocity vector plots at X/D = 0.30, and which is also indicated by the vorticity contours at this same location (Fig.4.6.2(a)). In comparison with a single jet, the magnitude of the bound vorticity is therefore much higher but is concentrated in a smaller area. The vorticity contours at X/D = 0.30 also appear to indicate the presence of the less pronounced counter-rotating horseshoe vortex system positioned close to the injection wall. Downstream of the injection plane cross-flow is drawn into the wake region behind each jet so that mass continuity dictates a decrease in the axial velocity component. Hence the compressional strain rate (∂U/∂X<0) results in a decrease in streamwise vorticity, the magnitude of which can be estimated from the mean velocity data.
Results and Discussion

For example, the axial velocity components at the centre of the bound vortex on the left side ($\theta > 0^\circ$) of hole 8 are 35 m/s and 22 m/s at $X/D = 0.30$ and 2.0 respectively. Using the substantial derivative form of eqn.(3.3.2) and assuming a mean axial velocity of 28.5 m/s between these locations there is approximately a 45% reduction in streamwise vorticity purely due to the compressional strain rate in the flow field at the centre of the bound vortex. This is the main reason for the rapid decay of the bound vortex system in the multiple jet configuration, and has important consequences when considering the design of the dilution annulus. By maintaining axial velocities in the downstream flow field the negative vorticity production can be minimised so enhancing mixing, and this explains the findings of Srinivasan et. al.\cite{21} concerning the favourable effects of introducing mixing duct convergence.

Of equal importance in terms of vortex decay is the diffusion of vorticity which is present in both single and multiple jet configurations, although the dissipation is enhanced by the greater velocity gradients in the latter case. Since the compressive strain rate is approximately the same either side of the centre-plane the asymmetry in the bound vortex system at $X/D=2.0$ (Fig.4.6.2(b)) is thought to be mainly due to differences in the diffusion of vorticity associated with variations in the trajectory of jet fluid. As vorticity reflects gradients in momentum which are eroded by turbulent stresses, Reynolds' analogy\cite{53} therefore suggests that a greater rate of diffusion reflects higher mixing so producing a more rapid rate of cooling on the right ($\theta > 0^\circ$) of the hole 8 centre-plane.

4.6.2 Cross-flow vortex system

Measurements conducted at $X/D=2.0$ indicate the complex nature of the flow field in comparison with that of the relatively simple bound vortices observed downstream of a single jet. In this case the bound vortex system has almost decayed and, although data is limited due to the finite probe calibration range, there does appear to be a concentration of streamwise vorticity located in the wake of the jet. Radial components of vorticity ($\Omega_y$) must be generated by the lateral shearing between the jet and cross-flow (Fig.4.6.3) and further downstream by the movement of flow into the separated wake region behind the jet, these effects being enhanced by the surrounding velocity field in a multiple jet configuration. As outlined by Sykes et. al.\cite{17} for a single jet, diffusion at the sides of the jet reduces this vorticity component, whilst behind the jet viscous effects are relatively small and the component is increased due to an extensional strain rate as fluid in the wake is entrained upward with the jet. This is represented by the $\Omega_y \partial U/\partial Y$ term in equation (3.3.2), and the convection of this vorticity into the axial direction is thought to be a major factor in producing the high regions of vorticity observed in the wake at $X/D=2.0$. 

55
Relative to a single jet in an unconfined cross-flow, the radial vorticity component at the injection plane will be higher due to the acceleration of fluid between the jets, and this effect is accentuated if the velocity gradient ($\partial U/\partial Y$) in the wake of each jet is greater in the confined cross-flow case. Furthermore, the empirical equations of Srinivasan et. al.[20] and Kamotani and Greber[5] outlined in section (1.2) indicate the significant reduction in penetration for the multiple jets relative to a single jet. (from $Y/D=1.83$ to 1.44 at $X/D=2.0$)

The influence of the injection plane wall (Fig.4.6.4) increases the velocity gradients associated with the fluid moving laterally into the wake region of each jet, thereby further increasing the radial vorticity component which is convected into the downstream direction in the jet wake.

The effect of the injection plane geometry can be illustrated by blanking off alternate holes on the test rig so that the space/diameter ratio ($S/D$) is increased from 2.75 to 5.50. By maintaining a momentum flux ratio of 4, the velocity distributions for both $S/D$ ratios can be compared and these indicate a change in the downstream flow field (Fig.4.6.5). Although the increase in jet penetration is associated with the decreased blockage in the cross-flow presented by only 8 dilution holes supplying fluid, there is also a change in the velocity components around the periphery of the jet. When all 16 jets are present a stronger downward movement of fluid occurs around the sides of each jet and into the wake region, probably leading to a stronger cross-flow vortex system developing. These observations are complicated however by the distortion of each jet, and the larger wake and more pronounced influence of the outer annulus wall associated with the increased jet penetration at the larger $S/D$ ratio. However, these results clearly indicate how changes of the injection plane geometry influence the downstream velocity distribution. It therefore follows that for closely spaced multiple jets issuing into a confined cross-flow it is to be expected that the downstream vortex flow fields will be different from that of a single jet issuing into an unconfined cross-flow.

4.7 Major factors influencing the dilution hole exit velocity profile

As the distortion of the downstream temperature patterns is associated with the vortices and other flow features issuing from the dilution holes, it is necessary to consider the origins of these non-uniformities and thereby assess how representative they are of the flow fields found in an operational combustor. Furthermore, it may be possible to modify the feed annulus and eliminate the non-uniformities issuing through the holes, thereby improving the regularity of the temperature distribution around the dilution annulus.

Some experimental work was conducted with the objective of showing how a well defined vortex such as that issuing from hole 10 can be formed. Very little information is
vortices in the dilution holes are formed by air from the feed annulus boundary layer.
Results and Discussion

available on how fluid passes through a dilution hole from a feed annulus, but recirculation of the flow must take place from downstream of the injection plane. Such effects together with an unstable region located in this recirculation zone have been observed in a Rolls-Royce RB211 low pollution research combustor that is mounted in a water analogy rig at Loughborough (Fig.4.7.1). This recirculation may not be symmetric about the hole centreline and its instability may mean that each hole is fed preferentially from one side or the other, and could result in vortices and other flow features issuing through the holes. The effects of preferentially supplying a hole from one side can be investigated byblanking off a single jet on the test facility, thereby producing asymmetric feed conditions to the adjacent holes. Dilution hole 6 was therefore blanked off and for reasons of convenience the subsequent tests were conducted with no cross-flow present and the outer casing removed. The vector plots (Fig.4.7.2) indicate how vortices can be induced into the adjacent preferentially fed holes (ie. numbers 5 and 7) which originally had exhibited no such flow features. Furthermore, the vortex that is observed in hole 7 induces an additional vortex but of opposite rotation in hole 8. This indicates how flow disturbances in one part of the test facility can be transferred around the annulus and affect the mixing in other sectors. However, the velocity vectors for hole 4 indicate that, despite carrying out a substantial geometric change to the test rig (ie. blanking off a complete hole), this is not sufficient to overcome the vortex which normally issues through this hole. Instead, a highly fluctuating time dependent flow pattern is observed issuing through the rear of the hole. Due to the uniform approach conditions and high tolerances to which the rig is constructed it must therefore be concluded that any preferential feed effects to a hole are small, and this cannot account for the vortices which occur under normal rig running conditions.

Flow visualisation methods using smoke and wool tufts were used to investigate the deflection of air as it passes from the feed annulus and through the dilution holes. As was the case above, these tests were conducted with the outer casing removed since vortices had been recorded across the exit plane of the dilution holes with no cross-flow present. The tests revealed that the vortex fluid originates from the boundary layer associated with the inner wall of the feed annulus. These observations are confirmed by the temperature distribution across the exit plane of hole 10, where due to heat transfer the relatively cooler fluid originating from the boundary layer of the inner casing produces a reduction in the measured temperature (Fig.4.7.3). Since the dilution air passing down the feed annulus must eventually separate from the inner wall, local disturbances and imperfections in the casing surface will mean the separation line will vary around the annulus. Such variations may result in the flow rolling up into a vortex or produce some of the other flow features observed issuing through the holes. These effects would be enhanced by the stretching of the vortex lines which occur due to the acceleration of the air as it passes from the feed annulus (≈7m/s) and through the dilution holes (≈29.5m/s).
Results and Discussion

The instability of the inner wall boundary layer can be inferred from a simple, 2-dimensional consideration of the velocity profile and curvature of the flow as it passes from the feed annulus and through the dilution hole. It was Rayleigh in 1916 who first proved by energy considerations that a steady inviscid, axi-symmetric curved flow is unstable to infinitesimal disturbances if the angular momentum decreases with increasing distance from the centre of curvature. A simplified 2-dimensional consideration of the flow feeding into a dilution hole indicates that although the flow feeding through the front half is stable, the decrease in velocity associated with the boundary layer on the inner casing wall means this fluid is unstable as it passes from the feed annulus and through the rear of the hole (Fig.4.7.4). Kármán described this effect qualitatively in 1934 which is outlined by Stuart in a chapter of Rosenheads[54] publication. When fluid elements are moving in circular paths the centrifugal force is balanced by a pressure gradient to maintain the trajectory (Fig.4.7.5). However, if a particle is displaced to a different radius then its new velocity, which can be calculated by assuming constant angular momentum, may result in a discrepancy between the centrifugal force and the pressure gradient required to maintain the new trajectory. Thus, it can be shown that for 2 radial locations \( r_1 \) and \( r_2 \) the flow is stable if \( (r_2u_2^2) > (r_1u_1^2) \) since the displaced particle is forced back to its original position. However, if \( (r_2u_2^2) < (r_1u_1^2) \) then instability arises and a displaced particle will move even further away from its original trajectory. Bradshaw[55] gives a theoretical description of this phenomena which includes the effects of viscosity which stabilises the flow to a limited degree, and also included in this publication is a review of relevant experimental work. For example, a boundary layer on a concave surface is subjected to destabilising effects due to streamline curvature and in certain regimes where the flow is laminar Görtler vortices can be formed (Fig.4.7.6). In addition, high levels of turbulence may also be generated in flows subjected to these destabilising effects which could also produce unusual features in the flow field.

In the case of a row of dilution holes the acceleration of the fluid from the feed annulus, the radial convergence of the flow into the hole and the recirculation of flow from downstream of the injection plane are significant effects which may be enhanced by the instability of the inner wall boundary layer due to streamline curvature. It is therefore not surprising that vortices and other flow features are observed issuing through the dilution holes, with no 2 holes exhibiting the same flow pattern. This is also commented on by Lefebvre[44], who indicates what effect combustor geometry can have on vortex formation:

"If the pitch of the dilution holes is greater than the annulus height, a vortex can form in the flow entering the hole; this changes the penetration and mixing characteristics of the dilution air jet. The strength of the vortex depends on the ratio of the annulus area, as measured in the plane of the holes, to the hole area. A high value of this ratio inhibits vortex formation."
These limits reflect the trends found by McGuirk\cite{56} at Imperial College, where vortices were observed in dilution holes when the feed annulus height was reduced by using different pressure casings. In addition, these trends to some extent reflect the observations made on this test facility concerning the formation of vortices. The ratio of the annulus height to the hole pitch reflects the velocity profile in the feed annulus and the influence of the inner wall boundary layer on the flow feeding through each hole. Furthermore, smaller feed annulus heights infer larger approach velocities thereby increasing the size of the recirculation region downstream of each hole. Since the pitch of the holes (S/D) on this test facility is 2.75 and the feed annulus height (H/D) is 1.41, it follows from Lefebvre's criterion that vortices can be expected to form in the dilution holes. The fact that vortex formation is inhibited by increasing the ratio of annulus area to hole area can be demonstrated by blanking off alternate holes around the test facility. For the holes on which measurements were made (Fig.4.7.7), the vector plots indicate that the strong, clearly defined vortices have been eliminated although smaller vortices are still present towards the centre of each hole. The downstream temperature distributions however for this configuration still indicate significant distortions of the temperature contours (Fig.4.7.8) which confirms the observations outlined in section (4.6). Although a strong clearly defined vortex will distort a jet, smaller vortices such as those issuing from hole 8 and which are also observed when alternate holes are blanked off can have a similar effect. Both types of flow features introduce momentum and trajectory variations and alter the mixing characteristics of each jet.

Only a limited amount of data is available concerning the velocity profiles across the exit planes of dilution holes in operational combustors. As indicated by Gradon and Miller\cite{57} for the tubo-annular Spey combustor though, problems were encountered due to the formation of vortices which altered the penetration and mixing of the dilution jets. In this case the effects were subdued by fitting a longitudinal splitter plate across each dilution hole along with a downstream 'dam' (Fig 4.7.9) which prevents the random recirculation of fluid in the feed annulus. Such flow control features can also be observed in several other combustors but their effectiveness is limited due to the distortion of the flame tube during its operational life. In particular, if the splitter becomes offset to one side of a hole it may promote, rather than inhibit, the formation of irregularities in the holes as indicated by Wilson\cite{38}. Although a combustor can be designed to a geometry which inhibits the formation of some of these features to a limited degree, other considerations such as the available liner pressure drop and the required air velocity for convective cooling of the flame tube means this is not always possible. Furthermore, it is thought only with a plenum feed could a uniform momentum distribution be obtained across the exit plane of each hole. This is indicated by the work of Moys and Stevens\cite{39} at Loughborough who obtained good regularity of the temperature pattern around the annulus when the row of holes were supplied from a feed annulus. In
addition, Shaw\textsuperscript{[33]} in his work on a Viper annular combustion chamber also found the temperature irregularities around the annulus were minimised with a plenum feed (section 1.2). This further highlights the need for representative feed conditions to the dilution holes when investigating the consistency of dilution jet mixing. Furthermore, to ensure repeatability of results it also follows that the location of the dilution holes must be kept fixed with respect to the inner wall of the feed annulus since rotation of either casing will affect the flow fields. Indeed, tests have indicated that relative rotation of either casing by as little as 2° on the Loughborough rig is sufficient to significantly alter the flow patterns issuing through the holes.

The geometry of a combustor means that boundary layers will always be present in the feed annulus, and the deflection of the flow as it passes through the dilution hole will result in the formation of vortices and other flow irregularities. Local variations due to manufacturing tolerances and combustor distortion mean that the exit plane velocity profile across each hole will vary from one jet to another, so leading to irregularities in the temperature patterns around the dilution annulus. In addition, the temperature patterns are not only extremely difficult to predict, but will vary between combustors which are built to the same nominal design. Any solution to the problem must therefore accept the upstream velocity profile in a feed annulus and the subsequent flow deflection into a hole, and so this present investigation has therefore concentrated on designing a dilution hole which makes the subsequent development of a jet insensitive to the feed annulus effects.

4.8 Modified dilution hole designs

The mechanisms by which the cross-flow and jet fluids mix are significantly modified by the velocity profile across the exit plane of a dilution hole. The complex flow field issuing through the rear of a hole influences the trajectory of the deflected jet fluid either side of the centre-plane and the development of the bound and cross-flow vortex systems at the lateral edges of the jet. Each jet therefore has its own individual mixing characteristics which are related to its exit velocity profile, giving rise to the observed distortions of the kidney shaped temperature profiles which vary both in magnitude and direction from one jet to another. To improve the temperature pattern around the annulus it is necessary to overcome the effects produced by the feed conditions to the dilution holes, so that each jet has virtually the same mixing characteristics in the dilution annulus. As indicated by Carrotte and Stevens\textsuperscript{[59]}, to make the development of a jet insensitive to approach flow effects it is necessary to control the trajectory of the fluid issuing through the rear of the dilution hole whilst also influencing the development of the downstream flow field. The above authors have already put forward several suggestions as to how this might be achieved\textsuperscript{[60]}. 

Results and Discussion
The present test facility limits the different types of dilution hole that can be tested since, due to difficulties in manufacturing new casings, any modifications have to be incorporated into the existing casings. Thus, the 2 types of modified dilution hole tested (Fig.4.8.1) were designed with these restrictions in mind, and were obtained by using the existing casing containing the standard plunged holes which were re-contoured using cellulose filler as a lining material. Although this means that the modified designs are subjected to similar approach and upstream boundary layer conditions that were present for the standard hole geometry, several disadvantages are introduced by using this method of construction. For example, it should be noted that each hole had to be individually shaped to obtain the desired profile, and so no 2 holes are identical. Although this is more representative of an engine where manufacturing tolerances and distortions occur, it does mean that the results from the test rig are pessimistic relative to the standard design incorporating virtually identical holes. Furthermore, the changes introduced by the modified hole designs mean that several geometrical parameters vary, which can also affect the mixing characteristics of the jets. For example, a change in area relative to a standard hole of 82% and 96% is introduced by Designs A and B respectively. Although the associated variation in mass flow is accounted for in the data analysis it does mean there is a change in the effective location at which measurements are made, since the physical distance (X) of the traverse station downstream of the injection plane is fixed. Thus, changes in the effective hole diameter mean the plane X/D=2.0 corresponds to an effective location of X/D_e=2.20 and 2.04 respectively for Designs A and B. No attempts are made to correct for this since it is assumed that changes in the distortions of the temperature patterns are negligible over such small distances. Furthermore, as outlined by Lefebvre\cite{44}, the decrease in hole area with respect to the area of the feed annulus can inhibit vortex formation and so further changes in the jet mixing characteristics may be introduced which cannot solely be attributed to the modified hole geometry. However, it should be remembered that large temperature distortions were still recorded even when much larger changes in the total dilution hole area were introduced by blanking off alternate holes.

4.8.1 3 Hole Sector Tests

Initial tests were undertaken with only 3 out of the 16 holes modified to the selected designs with the mixing characteristics of the jet at the centre of the sector being investigated. For the purpose of comparison, the same 3 holes were modified in each test and compared with that of the results obtained for the standard geometry.
a) Design A

In addition to the tests conducted on the complete design, the individual modifications to the standard hole were tested to assess their contribution to the overall performance. Thus, a standard hole with back-stop was tested as was a plunged hole incorporating a step change in radius \( R = 0.4D \) around the rear 180° of the hole. The temperature contours (Fig.4.8.2) indicate a reduction in the distortion of the temperature pattern associated with these modifications which can also be quantified using the jet distortion parameter (Fig.4.8.3).

Incorporating a step change in radius at the lateral edges of the hole is designed to trip the bound vortices which are known to develop at this location. In addition, the fluid issuing through the rear of the hole has less influence on the lower momentum fluid at the sides of the jet. The hot core formed by the fluid issuing through the rear of the hole however is still distorted (Fig.4.8.2(b)). This is virtually eliminated when a small (0.3D) backstop is tested with a standard plunged hole (Fig.4.8.2(c)), so that deflected jet fluid must pitch over or yaw around this component, as illustrated by the velocity distribution at \( Y/D = 0.35 \) (Fig.4.8.4). This dominates any influence on fluid trajectory that the complex flow field has, the location of which has been centralised at the rear of the jet by the back-stop. Since this component is present only around the rear 140° of the hole, hot jet fluid can be deflected into the vortex development region either side of the jet and therefore assists in setting up a downstream vortex flow field which is symmetric about the hole centre-plane. This is indicated by the temperature distribution in the region of vortex controlled mixing close to the injection wall (Fig.4.8.5). Several different lengths of back-stop were tested, and to have the required influence on jet mixing at the operating conditions stated the back-stop must not be less than 0.2D, with a nominal value of 0.3D being selected for the tests reported here. Further increases in length result in greater penetration of the jet into the dilution annulus without any significant improvement in the distortion parameter.

When the step change in radius and back-stop components are tested individually, an improvement in the jet mixing parameter is obtained when compared with the standard design (Fig.4.8.3). Combining these components to form design A produces a further reduction in temperature distortion, particularly close to the injection wall. Whereas for the standard design the maximum distortion of the jet about its centre-plane is associated with the hot core fluid \( r_n=0.4 \), this has been virtually eliminated by Design A with the peak distortion \( r_n=0.1 \) now being associated with the vortex controlled mixing region close to the injection wall.

The results indicate that these modifications have made the mixing characteristics of the jet less sensitive to the effects of the approach flow conditions.
Results and Discussion

The design philosophy for these modified holes (Fig.4.8.1) was to try and make the development of each jet less sensitive to the effects that are introduced when dilution fluid is supplied from a feed annulus. In particular, the complex flow field which issues through the rear of each annulus fed hole, and which varies from one hole to another, has a major influence on jet mixing. The modifications to the dilution holes therefore concentrate on controlling jet behaviour by trying to reduce the influence of this flow feature. For example, it has already been shown that lower momentum fluid at the front and sides of a jet takes on a more curved trajectory than that of the core, thereby forming the characteristic kidney shaped cross-section and vortex systems that are observed downstream of the injection plane. However, the deflection of this fluid differs from one side of the jet to the other due to the influence of the velocity profile across the rear of the hole. The step change in radius at the lateral edges of the modified holes and reduced diameter of the rear half of the jet is designed to try and minimise this effect. It is thought that the trajectory of the low momentum fluid passing around the sides of the modified jet should be less influenced by the flow issuing through the rear of the hole, and the sharp change in radius should also help to trip vortices of equal strength as they commence their development in this region. As the core of the jet is bent over by the cross-flow however, so the complex flow field issuing through the rear of the hole has a significant influence on the trajectory of the deflected jet core. The idea of a back-stop at the rear of a dilution hole is to try and control the trajectory of this deflected jet fluid and so reduce the influence of the flow at the rear of the jet. As the core of the jet is bent over by the cross-flow, deflected fluid has to pass over or yaw around the back-stop which represents a blockage to the flow and, if of suitable size, should be of greater influence than that of the complex flow field. Furthermore, by using a back-stop only over the rear 140° of the hole, fluid can spill around its sides and assist in setting up vortices of equal strength at the lateral edges of the jet.
b) Design B

The velocity measurements at \( Y/D = 0.05 \) have shown how jet fluid is deflected as it passes through a dilution hole due to the influence of the cross-flow. The zero plunging radius around the rear 180° of the hole is therefore designed to influence the way the jet is fed and the deflection of the fluid as it passes from the feed annulus into the cross-flow. Furthermore, the change in hole geometry at the lateral edges of the jet is designed to modify the vortex flow field which develops at this location.

The temperature distribution associated with this design indicates a distinct change in the mixing characteristics of the jet (Fig.4.8.6) and although some asymmetry is present there is still a significant improvement in the jet distortion parameter from that of the standard plunged hole (Fig.4.8.7). The bifurcated temperature distribution is thought to indicate an increase in the strength of the downstream vortex flow field, probably due to the generation of vorticity being concentrated immediately downstream of the step change in radius by the straight sides of the hole. Further downstream at \( X/D = 2.0 \) a comparison of the velocity vector plots and streamwise vorticity distributions produced by the Design B and standard hole configurations confirms these observations (Fig.4.8.8), although it should be remembered that the effective location (\( X/De \)) varies between the tests. However, the vector plots appear to indicate much stronger downward velocity components around the sides of the jet for the case of Design B, with much higher concentrations of vorticity being present in the jet wake.

4.8.2 8 Hole Sector Tests

Investigation of a single jet has indicated that changes can be made to its mixing characteristics by modifications to the geometry at the injection plane. These results however must be repeated by a number of holes to calculate the effect on the consistency of the temperature pattern around the annulus. Modifications were therefore carried out to 8 dilution holes with temperature measurements being made downstream of the 6 jets in the centre of the sector. The temperature patterns produced by design A (Fig.4.8.9) are compared with the standard geometry results presented earlier (Fig.4.8.10) for the same sector, and indicate the more uniform mixing characteristics of the modified dilution jets. It should be noted that this improvement has been achieved despite the method of modifying each hole individually which leads to some variation in the hole geometry around the sector. The maximum, block maximum and mean temperature distribution factors for both geometries indicate the change in the overall mixing characteristics for the modified Design A sector and the standard holes (Fig.4.8.11). As already outlined, this may be due to changes in hole area, space to diameter ratio (\( S/D \)) and differences in the effective downstream location (\( X/De \)) in addition to the effects of hole geometry changes. This should be noted when considering the mixing
Results and Discussion

The mixing performance can be quantified using the difference between the peak temperature at any radius around the sector and the associated block maximum (Fig.4.8.12), the Temperature Asymmetry Factor indicating a significant improvement when compared to the standard geometry. The repeatability of the test result was indicated by completing a further traverse at a single radial location \( r_n = 0.633 \) downstream of the Design A holes. A value of 5.14% was obtained which is in good agreement with the original value recorded of 5.01%. The improved overall mixing of the six jets is quantified using the standard deviation for the sector from that of its block distribution (eqn.3.2.5), the value of which has been reduced by approximately 50% across the annulus (Fig.4.8.13). As was indicated by the jet distortion parameter, the asymmetry associated with the core of each jet about its own centre-plane is reduced by the modified hole design and this leads to a significant improvement in the consistency of the temperature pattern around the sector. The maximum circumferential temperature deviation occurs at the same radial location \( r_n = 0.65 \) for the 2 types of hole being compared and is associated with differences in the overall penetration of each jet. It would therefore appear that the improved mixing characteristics of each jet core also produces a more uniform penetration of the jets. On the basis of these results it is intended that tests be conducted at Lucas Aerospace on this type of hole design, but only if these modified designs have similar discharge coefficients to that of a standard plunged hole.

Due to time constraints 8 hole sector tests incorporating design B were not conducted.

4.8.3 Hole Discharge Coefficients

The outer casing of the test facility was removed so that the jets effectively discharged into a plenum, with the discharge coefficients \( (C_d) \) of the new hole design being calculated from measurements made in the centre of a 3 hole sector. The mass-flow through a hole was obtained by traversing a 5 hole probe across the exit plane, although for design A this meant removal of the back-stop protruding from the injection wall. The mass-flow weighted pressure drop across the casing was obtained from a 5 tube pitot rake in the feed annulus, and so the hole discharge coefficients could be calculated using the formula:

\[
m_h = C_d A [2(p - p_a)]^{0.5}
\]

(4.8.1)

where \( P \) is the total pressure measured in the feed annulus and \( p_a \) the plenum pressure into which the jets discharge. To check the accuracy of the experimental method, results for standard holes (Table 4.8.1) are compared with the values obtained from the empirically based Rolls-Royce CODAS program for the same operating conditions and hole geometry. Although there is some discrepancy from this data, it is thought the experimental method is of
sufficient accuracy to indicate that the modified hole designs do not incur a significant penalty in terms of reduced $C_D$. The test data has indicated that discharge coefficients are sensitive to the leading edge hole profile where separations can take place. Since the flow entering the rear half of the hole is almost axial, modifications can be made to the hole geometry in this region which appear to have minimal effect on $C_D$ (Fig. 4.8.14).
University of Technology

LOUGHBOROUGH LEICESTERSHIRE LE11 3TU

Thesis for microfilming.

With Compliments

INTER LIBRARY LOANS
Chapter 5: CONCLUSIONS
Conclusions

A detailed experimental investigation has been conducted into the mixing characteristics of a row of 16 heated dilution jets injected into a confined annular cross-flow. The investigation has concentrated on the consistency of the temperature patterns around the annulus that are produced by the mixing of the jets with the cross-flow. The following conclusions have been drawn:

- A major influence on the mixing characteristics of the dilution jets is the way air is supplied to the dilution holes, and significant variations are recorded in the temperature patterns when the holes are supplied from a feed annulus.

- The deflection of the fluid as it passes from the feed annulus and through the dilution hole produces a complex flow field across the exit plane of each jet. Furthermore, the flow pattern issuing through a dilution hole can influence adjacent holes and so effects associated with one part of the annulus can be transferred to other sectors.

- The complex flow field issuing through a hole affects the mixing characteristics of the jet, altering the trajectory of the dilution fluid and the vortex flow field which develops downstream of the injection plane. This can result in an apparent 'twisting' or distortion of the temperature pattern about the hole centre-plane.

- The mixing of the dilution jets can be influenced by changes in geometry at the injection plane, with holes being modified to make the development of each jet less sensitive to the effects of the approach flow in the feed annulus. Controlling jet behaviour in this way can produce significant improvements in the consistency of the temperature patterns around the annulus.

- At the operating conditions tested, the modified hole designs exhibit virtually the same discharge coefficient values as that of standard plunged holes.

- The nature of this investigation has also revealed that there are significant differences between the structure of a single jet, issuing into a relatively unconfined cross-flow, and that of a multiple jet configuration in a confined cross-flow. In particular, there are significant differences in the vortex flow fields observed downstream of the injection plane.
Chapter 6: RECOMMENDATIONS FOR FURTHER WORK
The performance of both the standard and modified dilution holes needs to be assessed over a wider range of operating conditions, and only a limited amount of this testing can be undertaken on the existing facility. The extremely uniform approach conditions in both the cross-flow and feed annuli are not representative of those found in combustion chambers, and the sensitivity of the modified dilution holes to local asymmetries in the approaching flow therefore needs to be investigated. Wakes from burner feed arms can be simulated on the facility by placing struts in the feed annulus whilst localised changes in the height of this annulus can be used to assess the effects of flame tube distortion on jet behaviour. The influence of the bleed flow used for turbine blade cooling may also be of importance, since this may have a significant effect on the way air passes from the feed annulus and through each hole. This would prove difficult to accurately simulate though on the test facility in its present configuration. Furthermore, the performance of the modified holes may be dependent on the thickness of the injection casing and the jet to cross-flow momentum flux ratio at which the facility is operated. For example, at larger values of the momentum flux ratio it is thought the height of the back-stop must be increased to maintain control of the jet. Thus, the designs and modifications to the dilution holes presented in this thesis only demonstrate ideas and methods of controlling jet behaviour, and need to be developed further on a test facility which simulates the operating conditions of the specific combustor in which the modifications are to be incorporated.

Using the existing facility, it is thought that several new designs of hole could be developed which not only control jet behaviour but also influence the rate at which the cross-flow and dilution air mix. Although difficult to prove, it is thought that a strong vortex issuing through a dilution hole will improve the rate of mixing, and so hole designs may be possible which take advantage of this effect. This investigation has already shown how varying the pitch of the dilution holes can induce vortices across the hole exit planes. It may be desirable to deliberately induce vortices in this way and hence achieve an aerodynamically stable flow pattern with a possible additional advantage of enhanced rates of mixing. Although the subsequent jets which develop will have high degrees of temperature distortion, this may be acceptable if the level of distortion is predictable and can be designed for. In order to do this however, a detailed parametric study of the trajectory and behaviour of these jets would be required. In addition, numerous ideas have already been put forward\(^\text{60}\) as to how vortices may be generated at the injection plane so as to change jet behaviour in a suitable manner, and the existing facility is ideally suited to investigate such concepts.
References
References


References

13. Lineker, G.W., Private communication, 1987


References


34. Shaw, D., 'Mixing in an inclined annular combustor', Roll Royce internal report, No. GN24488, August 1981.


References


42. Wilson, C., 'An Investigation into the Mixing of Dilution Jets Discharging into a Confined Crossflow', Loughborough University, Dept. of Transport Technology, 1985.


45. MacDonald, N.H., 'Thermocouple type selection to suit the application', Control and Instrumentation, September 1978.


47. Wray, A.P., 'The Use of a 5-hole Probe as a Non-nulled Instrument and the Analysis of Test Data Using the Computer Program SHP1, SHP2 and SHP3', Loughborough University, Dept. of Transport Technology, Report TT86R02, 1986.
References


49. Wray, A.P., PhD thesis, to be submitted


58. Carrotte, J.F. and Stevens, S.J., 'Experimental studies of Combustor Dilution Zone Aerodynamics', AIAA Paper No. AIAA-88-3274 (see also 'Experimental Studies of Combustor Dilution Zone Aerodynamics Part II: Jet Development', accepted for publication, Jnl. of Propulsion and Power, Nov. 1990)

References


Tables
<table>
<thead>
<tr>
<th>$r_n$</th>
<th>Mean TDF</th>
<th>Max TDF</th>
<th>Block max TDF</th>
<th>Block centreline TDF</th>
<th>TAF (%)</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.018</td>
<td>0.099</td>
<td>0.204</td>
<td>0.179</td>
<td>0.179</td>
<td>2.57</td>
</tr>
<tr>
<td>0.083</td>
<td>0.139</td>
<td>0.302</td>
<td>0.253</td>
<td>0.217</td>
<td>4.95</td>
</tr>
<tr>
<td>0.167</td>
<td>0.250</td>
<td>0.464</td>
<td>0.400</td>
<td>0.252</td>
<td>6.48</td>
</tr>
<tr>
<td>0.250</td>
<td>0.291</td>
<td>0.562</td>
<td>0.477</td>
<td>0.271</td>
<td>8.43</td>
</tr>
<tr>
<td>0.333</td>
<td>0.303</td>
<td>0.595</td>
<td>0.509</td>
<td>0.294</td>
<td>8.61</td>
</tr>
<tr>
<td>0.417</td>
<td>0.320</td>
<td>0.589</td>
<td>0.514</td>
<td>0.387</td>
<td>7.48</td>
</tr>
<tr>
<td>0.500</td>
<td>0.316</td>
<td>0.619</td>
<td>0.568</td>
<td>0.564</td>
<td>5.09</td>
</tr>
<tr>
<td>0.583</td>
<td>0.208</td>
<td>0.533</td>
<td>0.481</td>
<td>0.481</td>
<td>5.20</td>
</tr>
<tr>
<td>0.667</td>
<td>0.067</td>
<td>0.252</td>
<td>0.200</td>
<td>0.200</td>
<td>5.11</td>
</tr>
<tr>
<td>0.750</td>
<td>0.008</td>
<td>0.040</td>
<td>0.023</td>
<td>0.023</td>
<td>1.73</td>
</tr>
<tr>
<td>0.833</td>
<td>0.002</td>
<td>0.011</td>
<td>0.004</td>
<td>0.001</td>
<td>0.68</td>
</tr>
</tbody>
</table>

Table 4.2.1. Temperature distribution factors ($X/D=2.0$)
<table>
<thead>
<tr>
<th>HOLE TYPE</th>
<th>$C_d$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Straight faced dilution hole ($t/D = 0.375$)</td>
<td>0.72 (0.69)</td>
</tr>
<tr>
<td>Standard plunged dilution hole ($R_p/D = 0.375$)</td>
<td>0.90 (0.91)</td>
</tr>
<tr>
<td>Design A</td>
<td>0.90</td>
</tr>
<tr>
<td>Design B</td>
<td>0.90</td>
</tr>
</tbody>
</table>

() Denotes values obtained using Rolls Royce CODAS program

Table 4.8.1. Dilution Hole Discharge Coefficients
Figures
a) Tubular

b) Tubo-annular

c) Annular

Fig. 1.1 Types of combustion chambers

A Compressor
B Combustor inner casing
C Combustor outer casing
D Flame tube
E Nozzle guide vanes
F Turbine
Fig. 1.2 Main features of a combustion chamber
\[
\frac{Y}{D} = 0.89 \, J^{0.47} \left( \frac{X}{D} \right)^{0.36} \quad \text{Velocity trajectory}
\]

\[
\frac{Y}{D} = 0.73 \, J^{0.52} \left( \frac{\rho_j}{\rho_c} \right)^{0.11} \left( \frac{X}{D} \right)^{0.29} \quad \text{Temperature trajectory}
\]

Fig. 1.3 Jet velocity and temperature centrelines - Kamotani and Greber\textsuperscript{[5]}
Fig. 1.4 Influence of jet angle on jet trajectory - Platten and Keffer[11]
Fig. 1.5 Development of a jet - Abramovich[3]
Fig. 1.6 Development of a jet - Andreopoulos and Rodi[^14]
Fig. 1.7 Integral prediction method for a single jet - Abramovich[3]
Fig. 1.8 Integral prediction method for a single jet - Adler and Baron[19]
Jet trajectories

Contours of constant velocity \((J = 15, X/D = 3.5)\)

Fig. 1.9 Measured and predicted velocity distributions - Adler and Baron\[^{[19]}\]
Fig. 1.10  Measured and predicted streamwise (U) velocity profiles - Sykes et al.\cite{17}
Fig. 1.11 Predicted temperature centrelines for single and multiple jets
Fig. 1.12 Temperature distribution produced by 45° angled slots - Srinivasan[22]
Fig. 1.13  Measured and predicted temperature distributions - Srinivasan et. al.\textsuperscript{[23]}
Fig. 1.14 Typical sector temperature pattern at exit of an annular combustor \((T/T_{\text{mean}})\)
Fig. 2.1(a) Test facility - working section
Exhaust duct to atmosphere

Top plenum chamber (19.6m³)

Inlet flare, air filter and honeycomb flow straighter

Feed annulus

Cross-flow annulus

Throttle

Mixed flow

Bottom plenum chamber (2.2m³)

--- Cross-flow air

--- Dilution air

Fig. 2.1(b) Test facility - airflow paths
Cross-flow air

Dilution air

\[ r_0 = 10D \]

\[ r_1 = 7D \]

\[ R_{y/D} = 0.375 \]

\[ S/D = 2.75 \quad H/D = 3.0 \]

Fig. 2.1(c) Test facility - injection plane geometry
a) Dilution hole insert

Plasticene fillet

Casing

View A-A

b) Insert mounted in casing

Fig. 2.2 Dilution hole manufacture
Fig. 2.3 Test facility - Bailey's\cite{41} and Wilson's\cite{42} configuration
Fig. 2.4  Thermocouple geometry
a) Cranked probe

b) Straight probe

Nominal dimensions only - geometry of individual probes can vary due to method of construction

Fig. 2.5 Five hole probe geometry
Fig. 2.6 Pitot probe rakes
Density of experimental data points per jet

Fig. 2.7 Data acquisition system - complete annular temperature survey
Spacer ring secured to thermocouple

Thermocouple positioned relative to centre casing by setting distance 'y'

Fig. 2.8 Method of thermocouple positioning - complete annular surveys
Density of experimental data points per jet

Fig. 2.9(a) Data acquisition system - sector temperature survey
Density of experimental data points per jet

Fig. 2.9(b) Data acquisition system - sector 5 hole probe survey
where:  
\[ h = C_p T \]  
and  
\[ m' = m_j' + m_c' \]

i.e.  
\[ m'T_{\text{meas}} = m_j'T_{j\text{meas}} + m_c'T_{c\text{meas}} \]

or  
\[ \frac{T_{\text{meas}} - T_{c\text{meas}}}{T_{j\text{meas}} - T_{c\text{meas}}} = \frac{m_j'}{m'} \]

**Fig. 3.1** Method of temperature correction
where:

\[ (p - p_{\text{ref}}) = (p - p_{\text{ref}})_{\text{meas}} \cdot \frac{q_{\text{ref}}}{q_{\text{meas}}} \]

and \( q_{\text{ref}} \) is the reference dynamic head to which the data is corrected.

Fig. 3.2 Method of pressure correction
Block distribution

where $J =$ nos. of jets

$$T_b = \frac{1}{J} \sum_{i=1}^{J} T_i$$

Fig. 3.3 Block temperature distribution
\[ \sigma_j = \frac{1}{(T_j - T_c)} \frac{2}{N} \sqrt{\frac{N/2}{\sum_{i=1}^{N/2} (T_{\varphi_i} - T_{-\varphi_i})^2}} \]

Fig. 3.4 Jet distortion parameter
Fig. 3.5 Presentation of cylindrical data grid
Velocity profiles for 3 equi-spaced circumferential locations

Inner wall boundary layer:

\[ \delta^* = 4.44 \text{ mm (0.175D)} \quad \theta = 3.40 \text{ mm (0.134D)} \quad H = 1.30 \]

Outer wall boundary layer:

\[ \delta^* = 1.30 \text{ mm (0.051D)} \quad \theta = 0.98 \text{ mm (0.039D)} \quad H = 1.33 \]

Fig. 4.1.1 Cross-flow annulus axial velocity profiles (X/D=-6.0)
Fig. 4.1.2  Feed annulus axial velocity profiles (X/D= -3.5)
Fig. 4.2.1 Radial variation of temperature factors (X/D=2.0)
Contour interval 0.05

Fig. 4.2.2 Temperature distributions downstream of holes 1 to 16 (X/D=2.0)
Contour interval 0.05

Fig. 4.2.3 Temperature distributions recorded by Wilson[42], \(X/D=2.0\)
Fig. 4.2.4 Temperature distributions recorded by Wilson\textsuperscript{[42]} (X/D=3.0)

Contour interval 0.05
Contour interval 0.05

Fig. 4.2.5 Temperature distributions recorded by Wilson\textsuperscript{[42]} (X/D=4.0)
Contour interval 0.05

Contours of constant temperature

Fig. 4.2.6  Temperature distributions downstream of holes 7 to 12 (X/D=2.0)
Fig. 4.2.7 Temperature distributions for holes 9 and 10
Fig. 4.3.1  Velocity distributions at the exit planes of holes 7 to 12 (Y/D=0.05)
- Without cross-flow
Fig. 4.3.2 Velocity distributions at the exit planes of holes 9 and 10 (Y/D=0.05) - Without cross-flow
Fig. 4.3.3 Velocity distributions at the exit planes of holes 9 and 10 ($Y/D=0.05$) - With cross-flow
Fig. 4.3.4 Pitch angle variation of the flow issuing through the dilution hole (Y/D=0.05)
Fig. 4.4.1 Velocity vectors downstream of holes 7 to 12 (X/D=2.0)
(a) Axial velocity distribution ($U/V_j$)

(b) Velocity vector plot

(c) Temperature distribution

Fig. 4.4.2(a) Velocity and temperature distributions
(Hole 8, $X/D = 2.0$)
(a) Axial velocity distribution \( \frac{U}{V_j} \)

(b) Velocity vector plot

(c) Temperature distribution

Fig. 4.4.2(b) Velocity and temperature distributions (Hole 10, X/D=2.0)
Fig. 4.4.3 Recirculating regions downstream of a jet in a cross-flow
Fig. 4.5.1 Velocity and temperature distribution
(Hole 8, X/D=0.30)
Fig. 4.5.2 Velocity and temperature distribution (Hole 8, X/D=0.54)
Fig. 4.5.3 Velocity vector plot
(Hole 8, Y/D=0.20)
Fig. 4.5.4 Streamwise \((U/V_j)\) velocity component profiles (Hole 8, \(Y/D=0.20\))
Fig. 4.5.5  Streamwise and circumferential velocity component profiles (Hole 8, X/D=0.54)
Fig. 4.5.6 Temperature distribution
(Hole 8, Y/D=0.20)
Fig. 4.5.7 Velocity and temperature distribution (Hole 8, X/D=2.0)
Fig. 4.5.8 Velocity vector plot
(Hole 10, Y/D=0.20)
(a) Axial velocity distribution ($U/V_j$)

(b) Velocity vector plot

(c) Temperature distribution

Fig. 4.5.9 Velocity and temperature distribution (Hole 10, $X/D=0.54$)
Fig. 4.5.10  Velocity and temperature distribution (Hole 10, X/D=0.72)
(a) Axial velocity distribution ($U/V_j$)

(b) Velocity vector plot

(c) Temperature distribution

Fig. 4.5.11 Velocity and temperature distribution (Hole 10, $X/D=2.0$)
Fig. 4.5.12  Temperature distribution
(Hole 10, Y/D=0.20)
Fig. 4.6.1 Characteristics of multiple jets in a confined cross-flow
a) $X/D = 0.30$

b) $X/D = 2.00$

Fig. 4.6.2 Streamwise vorticity distribution ($\Omega_xD/V_j$) (Hole 8)
Fig. 4.6.3 Radial vorticity distribution ($\Omega_r D/V_f$) 
(Hole 8, X/D=0.30)
a) Single jet (J=4.0)

b) Multiple jet in a confined cross-flow (S/D=2.75, H/D=3.0, J=4.0)

Fig. 4.6.4 Influence of injection wall on the flow field downstream of a jet
a) Hole spacing to diameter ratio (S/D) = 2.75

b) Hole spacing to diameter ratio (S/D) = 5.50

Fig. 4.6.5  Velocity vector plot (X/D=2.0)
Feed annulus flow

Recirculation zone

Flow recirculation from downstream of the hole may not be symmetric about hole centreline

Fig. 4.7.1 Water analogy rig observations of flow feeding a dilution hole
Fig. 4.7.2  Velocity vector plots for holes 4 to 8 (Y/D=0.05)
- without cross-flow
Fig. 4.7.3 Flow visualisation of vortex formation
Fig. 4.7.4  Instability of flow feeding a hole
At radius $r_1$:

Angular momentum = $u_1 r_1$

Centrifugal force = $\rho \frac{u_1^2}{r_1}$

Force due to pressure gradient = $\rho \frac{u_1^2}{r_1}$

If particle is displaced to $r_2$:

Particle velocity = $u_1'$

For constant angular momentum $u_1' r_2 = u_1 r_1$ (ie. $u_1' = \frac{u_1 r_1}{r_2}$)

Centrifugal force = $\rho \frac{u_1'^2}{r_2} = \rho \frac{r_1^2 u_1^2}{r_2^3}$

Force due to pressure gradient = $\rho \frac{u_2^2}{r_2}$

If $(r_2 u_2^2) > (r_1 u_1^2)$ flow is stable - particle returns to its original path

If $(r_2 u_2^2) < (r_1 u_1^2)$ flow is unstable - particle deviates from original path

Fig. 4.7.5 Streamline curvature effects
Fig. 4.7.6 Formation of Görtler vortices on concave surface
a) Hole spacing to diameter ratio (S/D) = 2.75

b) Hole spacing to diameter ratio (S/D) = 5.50

Magnitude of resolved velocity component: $V_j$

Fig. 4.7.7 Velocity vectors for holes 9 and 11 $(Y/D=0.05)$ -without cross-flow
Fig. 4.7.8 Temperature distributions downstream of holes 9 and 11 (X/D=2.0)
Fig. 4.7.9 Flow control devices
Fig. 4.8.1 Modified dilution holes
Fig. 4.8.2 Temperature distributions - Design A components
(X/D=2.0)
Fig. 4.8.3 Jet distortions - Design A components
Fig. 4.8.4 Velocity vector plot for standard hole with back-stop $(Y/D = 0.35)$
Fig. 4.8.5 Temperature distributions
Fig. 4.8.6 Temperature distributions - Design B
(X/D=2.0)
Fig. 4.8.7 Jet distortion - Design B
Fig. 4.8.8 Comparison of standard and Design B downstream flow fields (X/D=2.0)
Fig. 4.8.9 Temperature distributions downstream of holes 7 to 12 (Design A, X/D=2.0)
Temperature distributions downstream of holes 7 to 12 (Standard holes, X/D=2.0)
a) Standard holes

b) Design A

Fig. 4.8.11 Temperature distribution factors (X/D=2.0)
Fig. 4.8.12 Temperature asymmetry factor for Design A and standard hole sectors
Fig. 4.8.13  Standard deviations for Design A and standard hole sectors
Fig. 4.8.14 Hole discharge coefficients
Appendix 1: VORTICITY
Vorticity

It is the purpose of this appendix to give a qualitative rather than a quantitative description of the concept of vorticity and relate this to the flow field observed downstream of a dilution jet. Many authors such as Massey\cite{61} offer a general description, whilst several workers such as Lighthill\cite{62} give a more detailed mathematical representation of vorticity and how this may be applied to various types of flow field. Very few derivations and proofs will be derived here, merely the consequences and meanings of results will be discussed which assist in describing the flow observed downstream of a dilution jet. For a more widely applicable and detailed description of vorticity it is suggested the above references be consulted.

The concept of vorticity is concerned with the angular velocity of a fluid element rather than its linear velocity component. The vorticity vector is defined in terms of the velocity \( \mathbf{V} \) as 

\[
\mathbf{Q} = \text{Curl} \mathbf{V}
\]

so that its \( X, Y \) and \( Z \) components are defined as:

\[
\begin{align*}
Q_x &= \omega \frac{\partial v}{\partial y} - \omega \frac{\partial u}{\partial z} \\
Q_y &= \omega \frac{\partial u}{\partial z} - \omega \frac{\partial w}{\partial x} \\
Q_z &= \omega \frac{\partial w}{\partial x} - \omega \frac{\partial v}{\partial y}
\end{align*}
\]

(A.1.1(a))

(A.1.1(b))

(A.1.1(c))

Related to the concept of vorticity is that of circulation which is defined as

\[
\text{Circulation } K = \oint V_s \, ds
\]

(A.1.2)

where \( V_s \) represents the component of velocity along an element of a closed circuit. If one considers an elementary 2-dimensional rectangle (Fig.A.1.1) of size \( \delta x \delta y \) then using equation (A.1.2)

\[
\text{Circulation } K = u \delta x + \left( v + \frac{\partial v}{\partial x} \delta x \right) \delta y - \left( u + \frac{\partial u}{\partial y} \delta y \right) \delta x - \mathbf{V} \delta y
\]

or

\[
K = \left( \frac{\partial v}{\partial x} - \frac{\partial u}{\partial y} \right) \delta x \delta y
\]

(A.1.3)

where \text{Area of element} = \delta x \delta y
Thus, it can be seen that the vorticity is defined as the ratio of the circulation around an infinitesimal circuit to that of the area of the area of that circuit.

In a direct analogy to how velocity vectors can be joined to form streamlines, so the local vorticity vectors in a flow field can be joined up to form vortexlines (Fig.A.1.2). Furthermore, a vortextube can be defined as a surface comprising of vortexlines passing through a particular loop which is similar to that of a streamtube in a velocity field. The development of these features in a flow field and their relationship with the fluid motion is described by 2 theorems:

(i) Kelvins theorem: The circulation around a closed curve moving with the fluid remains constant.

(ii) Helmholtz theorem: Vortexlines move with the fluid.

Both of these theorem are only exact for the Euler model where viscous forces are neglected.

Since the limit of a vortextube as its cross-section is reduced to zero represents a vortexline, so these features have similar properties and Helmholtz theorem also means that vortextubes move with the fluid. Thus, particles of fluid of which a vortexline (or vortextube) is composed of move in such a manner that the same chain of particles continue to form a vortexline at later intervals of time. Furthermore, as outlined by Lighthill[62], a closed curve moving with the fluid can be thought of as a necklace of fluid particles whose shape and position change continuously as the particles of fluid move. It must therefore be concluded from these theorems that vortexlines (or vortextubes) move with the fluid and that the circulation is constant. Thus, as a vortexline is stretched due to acceleration of the fluid so the cross-sectional area will decrease and as indicated by eqn.(A.1.3) the vorticity magnitude will increase. Hence, vorticity magnitude increases or decreases in proportion to the stretching or shortening of the chain of particles which form the vortexline. In a 3-dimensional flow field the direction of the vorticity vector will also vary as the direction of the chain of particles changes as it moves with the fluid.

These properties are evident in the equations which define the rate of change of a vorticity component with time. These are derived by ‘taking the curl’ of the Navier Stokes equations and manipulating the subsequent formulae. For example, the \(x\) component is given as

\[
\frac{\partial \Omega_x}{\partial t} + U \frac{\partial \Omega_x}{\partial x} + V \frac{\partial \Omega_x}{\partial y} + W \frac{\partial \Omega_x}{\partial z} = \Omega \frac{\partial U}{\partial x} + \Omega \frac{\partial U}{\partial y} + \Omega \frac{\partial U}{\partial z} + \nu \left( \frac{\partial^2 \Omega_x}{\partial x^2} + \frac{\partial^2 \Omega_x}{\partial y^2} + \frac{\partial^2 \Omega_x}{\partial z^2} \right)
\]
Vorticity

This can be written in substantial derivative form as

\[
\frac{\partial \Omega_x}{\partial x} = \left( \Omega_x \frac{\partial U}{\partial x} + \Omega_y \frac{\partial U}{\partial y} + \Omega_z \frac{\partial U}{\partial z} \right) + u \left( \frac{\partial^2 \Omega_x}{\partial x^2} + \frac{\partial^2 \Omega_y}{\partial y^2} + \frac{\partial^2 \Omega_z}{\partial z^2} \right) \quad (A.1.4)
\]

This represents the rate of change of the x component of vorticity of a fluid element as it moves along a streamline. The first group of terms on the right of equation (A.1.4) represents the change in vorticity due to the stretching of the vortexlines. Thus, the term \( \frac{\partial U}{\partial X} \) indicates the direct change due to the elongation or shortening of the X component of vorticity, whilst the other 2 terms represent the turning of the Y and Z components of vorticity into the X direction. The Kelvin and Helmholtz theorems are only exact where viscous forces are not present, and the second group of terms on the right of equation (A.1.4) represents the diffusion of vorticity which takes place due to the viscosity of a fluid. Thus, there is a change in vorticity due to viscous action which can be thought of as an effective thickening or increase in cross-sectional area of the vortex tubes (or vortexlines) as they move with the fluid.

Vorticity is created in a number of ways and its subsequent convection and diffusion in the flow field obeys that dictated by equation (A.1.4). For example, the behaviour of a boundary layer can be described in terms of the vorticity created at a solid surface and how the vortexlines are then convected and diffused with the fluid. For a dilution jet vorticity is created by the jet as it issues into the cross-flow, and the subsequent movement of the fluid particles has a significant effect on the downstream flow field and the mixing of the air streams. Sykes[17] using a computer model calculated the vorticity field and was able to draw vortexlines in regions of significant vorticity. The trajectory of these vortexlines result in the creation of 'line vortices' downstream of the injection plane. These are large concentrations of vorticity around single lines and are the counter-rotating vortices that have been measured by many workers downstream of the injection plane. It is these vortices which have a significant influence on the mixing of the cross-flow and dilution air streams.
Mean velocity along $AB = U + \frac{\partial U}{\partial X} \cdot \frac{dX}{2}$

Mean velocity along $BC = V + \frac{\partial V}{\partial Y} \cdot \frac{dX}{2} + \frac{\partial V}{\partial Y} \cdot \frac{dY}{2}$

Mean velocity along $DC = U + \frac{\partial U}{\partial X} \cdot \frac{dX}{2} + \frac{\partial U}{\partial Y} \cdot dY$

Mean velocity along $AD = V + \frac{\partial V}{\partial Y} \cdot \frac{dY}{2}$

Circulation $(K) = U_{AB} \cdot dX + V_{BC} \cdot dY - U_{DC} \cdot dX - V_{AD} \cdot dY$

Fig. A.1.1 Definition of circulation
a) Vorticity vectors joined to form vortexlines

b) Vortexlines forming a vortextube

Fig. A.1.2 Vortexlines and vortextubes
Appendix 2: NON-NULLED FIVE HOLE PROBE THEORY

A.2.1 Theory

A.2.2 Calibration

A.2.3 Using the five hole probe

Figures
The convention for hole numbering and the various angle definitions are given in Fig.A.2.1, and the associated nomenclature which is used in this section is as follows:

\[
\begin{align*}
P_n & \quad \text{pressure sensed by one of the five holes} \\
p_i & \quad \text{pressure sensed by one of the side holes (2 or 4)} \\
K_n & \quad \text{fractional part of flow dynamic head which is sensed by hole n} \\
X & \quad \text{pitch parameter} \\
Y & \quad \text{yaw parameter} \\
D_p & \quad \text{dynamic pressure parameter} \\
S_p & \quad \text{stagnation pressure parameter} \\
\text{PTR} & \quad \text{true pitch angle} \\
\text{PPS} & \quad \text{pseudo pitch angle} \\
\text{YTR} & \quad \text{true yaw angle} \\
\text{YPS} & \quad \text{pseudo yaw angle}
\end{align*}
\]

A.2.1 Theory

When a probe is aligned approximately in the flow direction, each of the holes register a pressure \( p_n \) which is the sum of the local static pressure \( p \) and a proportion of the dynamic head \( q \) i.e.

\[
p_n = p + K_n q \tag{A.2.1}
\]

As Mach number and Reynolds number effects can be considered negligible, the value of \( K_n \) is dependent only on the direction of the flow relative to the hole. This relationship is used as the basis for the definitions of \( X, Y, D_p, \) and \( S_p \).

The pitch \( (X) \) and yaw \( (Y) \) parameters are defined as:

\[
\begin{align*}
X &= \frac{p_1 - p_3}{p_5 - p_1} = \frac{K_1 - K_3}{K_5 - K_1} \tag{A.2.2} \\
Y &= \frac{p_2 - p_4}{p_5 - p_1} = \frac{K_2 - K_4}{K_5 - K_1} \tag{A.2.3}
\end{align*}
\]

These two parameters are non-dimensionalised by a function of the dynamic head \( (p_5 - p_1) \) and are therefore independent of the flow velocity. Thus, for any particular flow direction there is a unique \( (X, Y) \) pair provided that the flow angle does not exceed approximately 36° since, beyond this, holes can lie in the wake from the probe tip and equation (A.2.1) is no longer applicable. The particular hole denoted by subscript \( i \) is taken
as either hole 2 or 4 depending on which gives the largest value of \((p_5 - p_i)\) in order to maintain the range of \(X\) and \(Y\) values sensibly low.

For any given flow direction, the difference between the pressure sensed by the centre hole and any of the other four must be a function of flow velocity. Using equation (A.2.1) we obtain:

\[(p_5 - p_i) = q(K_5 - K_i)\]

By defining \(D_p\) as \((K_5 - K_i)\) we have a dynamic pressure parameter which, like \(X\) and \(Y\), is a function of flow direction only. Thus:

\[q = \frac{p_5 - p_i}{D_p}\]  \hspace{1cm} (A.2.4)

Considering flow along an incompressible streamline, then

\[P = p + q\]

Also, considering hole 5, equation (A.2.1) gives

\[p = p_5 - K_5 q\]

and by eliminating \(p\) from the above equations we obtain

\[(P - p_5) = q (1 - K_5)\]

and non-dimensionalising:

\[\frac{P - p_5}{p_5 - p_i} = \frac{1 - K_5}{K_5 - K_i}\]

The stagnation pressure parameter \(S_p\) is defined as

\[S_p = \frac{1 - K_5}{K_5 - K_i}\]  \hspace{1cm} (A.2.5)

so that the following expression can be used to obtain the stagnation pressure:

\[P = p_5 + S_p (p_5 - p_i)\]  \hspace{1cm} (A.2.6)

### A.2.2 Calibration

The calibration procedure involves subjecting the probe to an air stream of known dynamic pressure over a range of pitch and yaw angles which cover all the variations of flow incidence angles likely to be encountered in service. A calibration traverse mechanism rotates the probe in two directions in order to present compound flow incidence onto the probe tip, and provides ranges of true yaw and pseudo pitch of \(+36^\circ\) to \(-36^\circ\) in increments of \(2^\circ\) or \(4^\circ\).

The calibration procedure is automated in a similar manner to the data acquisition system used on the test rig, with the data being transmitted from an LSI 11/23 microprocessor to a PDP 11/34 computer where it is stored on disk. A typical calibration file contains 361 data points,
with an (X, Y) pair being calculated at each of the angles tested. The data contained in the calibration file consists of the PPS, YTR, $D_p$ and $S_p$ for each (X, Y) pair.

### A.2.3 Using the five hole probe

When data has been acquired from the test facility the velocity, total pressure and dynamic head for each data point is derived as follows:

i) The X and Y parameters are calculated for each data point

ii) For each (X, Y) pair, the closest 25 points from the calibration file are selected (i.e. the closest 25 X,Y coordinates and their associated PPS, YTR, $D_p$ and $S_p$ values).

iii) A least squares bi-quadratic surface is fitted to each of the four parameter arrays (i.e. PPS, YTR, $D_p$ and $S_p$) and their values at the test (X, Y) coordinates calculated. Hence, the values of PPS, YTR, $D_p$ and $S_p$ are known for each data point.

iv) Using these parameter values together with $p_5$ and $p_1$ the true pitch, true yaw, stagnation pressure and dynamic head are calculated using equations (A.2.1) to (A.2.6)

The software used to perform the above analytical steps is complex and is described in greater detail by Wray[47].
Fig. A.2.1 Five hole probe geometry and nomenclature