A study of nozzle exit boundary layers in high-speed jet flows

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

Additional Information:

• A doctoral thesis submitted in partial fulfilment of the requirements for the award of Doctor of Philosophy at Loughborough University.

Metadata Record: https://dspace.lboro.ac.uk/2134/31797

Publisher: © M.T. Trumper

Rights: This work is made available according to the conditions of the Creative Commons Attribution-NonCommercial-NoDerivatives 2.5 Generic (CC BY-NC-ND 2.5) licence. Full details of this licence are available at: http://creativecommons.org/licenses/by-nc-nd/2.5/

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

For the full text of this licence, please go to:
http://creativecommons.org/licenses/by-nc-nd/2.5/
Please note that fines are charged on ALL overdue items.
A Study of Nozzle Exit Boundary Layers in High Speed Jet Flows

by

Miles Thomas Trumper, M.Eng.(Hons)

A Doctoral Thesis

Submitted in partial fulfilment of the requirements for the award of Doctor of Philosophy of Loughborough University 1st September 2006

© M.T.Trumper, 2006
Abstract

The requirement for reduced jet noise in order to meet stringent noise legislation (civil aviation), and low infra red observability and the use of unconventional exhaust nozzle configurations to improve aircraft survivability and performance (military aviation) is driving research to develop a better understanding of jet development and mixing mechanisms. One option open to the engineer is the use of small-scale model testing to investigate jets flows and provide valuable data for the validation of numerical models. Although more economical than large/full-scale testing, additional factors that influence jet development may be present which would not be present at full scale and whose influence needs to be fully understood in order to allow small-scale - large-scale read across. One such factor is the nozzle exit boundary layer. Although considerable data exist on the influence of nozzle exit boundary layers on low speed jet flows, current information on high speed jet flows is limited. It was, therefore, the aim of this thesis to extend the current understanding of nozzle exit boundary layers and their influence on the jet development for high speed jet flows through a combination of experimental and computational techniques.

A combination of pneumatic probe measurements and laser Doppler anemometry (LDA) was used to investigate nozzle inlet and exit boundary layers of simple conical nozzles and the influence of adding a parallel extension piece. The measurements showed that the rapid acceleration of the boundary layer within the nozzle significantly reduced its momentum thickness Reynolds number and changed the state of the boundary layer from turbulent to laminar-like. The addition of a parallel extension to the nozzle exit returned the boundary layer to a fully turbulent state.

A low Reynolds number RANS CFD approach was used to investigate the flow within the nozzle. Simulations using the Launder-Sharma low Reynolds number $k - \epsilon$ model revealed that the magnitude of the acceleration within the conical nozzles resulted in the boundary layer beginning to relaminarise. Full relaminarisation was not achieved due to the short axial distance over which the acceleration was sustained. The addition of a parallel extension provided a relaxation region in which the boundary layer could recover from the acceleration to become fully turbulent.

Measurements of the jet plume originating from nozzles with laminar-like and turbulent boundary layers showed little influence of the boundary layer shape and thickness on shear layer spreading and jet centreline development.

Keywords: Nozzles, Jet Plume, Initial Conditions, Boundary Layer, Relaminarisation, Compressible, Turbulence Modelling
Every man gets a narrower and narrower field of knowledge in which he must be an expert in order to compete with other people. The specialist knows more and more about less and less and finally knows everything about nothing.

*Konrad Lorenz*, Nobel Laureate (1903 – 1989)
Acknowledgements

Finally, the wider world has beckoned and this thesis marks the end of more than twenty-one years of formal education. Although only one name appears on the title page of this thesis, it would not have been possible without the support, guidance, understanding and perseverance of many others whom I have been most fortunate to have met throughout the previous three and a half years of study and to whom I am eternally indebted.

First and foremost, I would like to express my gratitude to Professor James McGuirk, who has tirelessly supervised this project. Apart from providing me with the opportunity to undertake this study, he has provided invaluable guidance, expertise and genuine enthusiasm every step of the way. Of equal importance was his remarkable ability to rekindle my enthusiasm in the project during the periods of doubt which were encountered en route. This thesis would also not have been possible without the support of Dr. Gary Page who provided the DELTA CFD code and considerable guidance and assistance on its use and modification. I am also most grateful for his support throughout the project as a whole. I would also like to acknowledge the help and guidance of Kimball Bradbrook, Steven Nunn and John Whitehouse of BAE SYSTEMS, Warton, for their valuable comments and suggestions on the direction of the project.

I would also like to offer my sincerest thanks to Dr. Parviz Behrouzi and Dr. Tong Feng for their assistance with the experimental measurements and the operation of the High Pressure Nozzle Test Facility, Dr. Dachun Jiang for his support and advice on the implementation of modifications to the DELTA code and Dr. Indi Tristanto for his advice on many numerical aspects of this thesis and his help with \LaTeX. No experimental project would be possible without the support of numerous technical staff. Particular gratitude is due to Mr David Roache for his help in producing nozzles, probes and a boundary layer traverse at the drop of a hat and Mr Paul Reeves for his assistance with all of the electrical aspects of the project. Special mention is needed for my dear friend and colleague Mr Ioannis Veloudis, with whom I have worked throughout the whole of this project and with whom I have shared advice, knowledge, ideas, despair and coffee.

Finally, I would like to acknowledge the financial support of BAE SYSTEMS and EPSRC grant number GR/02300852 without which the project would not have been possible.
Contents

Abstract ................................................................. II
Acknowledgements ...................................................... IV

1 Introduction .......................................................... 1

1.1 Introduction ....................................................... 1

1.1.1 Exhaust Nozzle Applications-Civil Aviation .............. 1

1.1.2 Exhaust Nozzle Applications-Military Aviation .......... 3

1.2 Experimental Studies of Nozzle Performance and Jet Plume Aerodynamics at Reduced Scale ................. 6

1.3 The Influence of Nozzle Exit Conditions on Jet Shear Layer Development ........................................ 8

1.3.1 The Influence of Jet Exit NPR or Mach Number, Jet Density and Jet Temperature .............................. 11

1.3.2 The Influence of Jet Reynolds Number ..................... 14

1.3.3 The Influence of Nozzle Exit Boundary Layers .......... 15

1.4 Factors Influencing Nozzle Exit Boundary Layers .......... 20

1.5 The Turbulent Zero Pressure Gradient Boundary Layer ................................................................. 21

1.5.1 The Inner Region ............................................... 21

1.5.2 The Outer Region .............................................. 23

1.6 Pressure Gradient Effects on Boundary Layers .......... 24

1.6.1 Mechanism of Relaminarisation ............................. 26
## CONTENTS

1.6.2 Behaviour of the 'Relaminarised' Boundary Layer ............... 28  
1.6.3 Previous Experimental Work on Relaminarising Boundary Layers ... 29  
1.6.4 Previous Numerical Work on Relaminarising Boundary Layers ..... 33  
1.7 Summary .................................................. 34  
1.8 Aims and Objectives ......................................... 36  
1.9 Thesis Structure ............................................. 37  

2 Experimental Facilities and Methodology .......................... 38  
2.1 The Test Facility ............................................ 39  
2.1.1 Air Supply .............................................. 39  
2.1.2 Air Supply Pressure and Mass Flow Rate Control ............... 40  
2.1.3 Combustion System ....................................... 41  
2.1.4 Air Delivery to the Test Nozzle ............................. 41  
2.1.5 Test Nozzle Design ....................................... 42  
2.2 Pneumatic Probe Measurements .................................. 43  
2.2.1 Common Instrumentation .................................. 43  
2.2.2 Nozzle Inlet Boundary Layer Measurement ..................... 44  
2.2.3 Nozzle Exit Boundary Layer Measurement ..................... 46  
2.3 Laser Doppler Anemometry .................................... 49  
2.3.1 LDA System: Configuration and Operation ..................... 50  
2.4 Data Reduction .............................................. 56  
2.4.1 Pneumatic Probe Measurements .............................. 56  
2.4.2 LDA Measurements ........................................ 57  
2.4.3 Derived Quantities ....................................... 58  
2.5 Error Sources ................................................. 64  
2.5.1 Pressure Measurements ..................................... 64
## 3 Numerical Modelling Methodology

3.1 Introduction .......................................................... 89
3.2 Governing Equations ............................................... 90
  3.2.1 Time-Averaging of the Navier-Stokes Equations .......... 92
  3.2.2 Constant Enthalpy Assumption ............................... 93
3.3 Turbulence Modelling ............................................... 93
  3.3.1 $k - \varepsilon$ Turbulence Model ......................... 94
3.4 Generic Transport Equation .................................... 97
3.5 Grid Generation and Geometry Definition .................... 97
  3.5.1 Geometry Definition .................................... 98
3.6 Boundary Conditions ............................................... 99
3.7 Solution Approach ................................................ 100
3.8 Code Validation .................................................... 101
  3.8.1 Low Reynolds Number Pipe Flow ....................... 101
  3.8.2 Relaminarising Axisymmetric Boundary Layer .......... 104
3.9 Closure ............................................................. 110
3.10 Figures ............................................................ 111

## 4 Experimental Results

4.1 Introduction ......................................................... 121
4.2 Nozzle Inlet Measurements (Pneumatic Probe Data) ............. 122
4.3 Nozzle Exit Measurements .......................................... 127
  4.3.1 Pneumatic Probe Data .................................. 127
<table>
<thead>
<tr>
<th>Section</th>
<th>Title</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>4.3.2</td>
<td>LDA Data</td>
<td>131</td>
</tr>
<tr>
<td>4.4</td>
<td>Influence of Addition of a Parallel-Walled Nozzle Exit Extension</td>
<td>137</td>
</tr>
<tr>
<td>4.5</td>
<td>Influence of Nozzle Exit Boundary Layers on Jet Plume Development</td>
<td>140</td>
</tr>
<tr>
<td>4.5.1</td>
<td>Near-field Development of Jet Plumes - Subsonic NPRs</td>
<td>140</td>
</tr>
<tr>
<td>4.5.2</td>
<td>Near-field Development of Jet Plumes - Supersonic (Underexpanded) NPRs</td>
<td>145</td>
</tr>
<tr>
<td>4.6</td>
<td>Closure</td>
<td>147</td>
</tr>
<tr>
<td>4.7</td>
<td>Figures</td>
<td>149</td>
</tr>
<tr>
<td>5</td>
<td>Numerical Prediction of Nozzle Boundary Layer Development</td>
<td>197</td>
</tr>
<tr>
<td>5.1</td>
<td>Introduction</td>
<td>197</td>
</tr>
<tr>
<td>5.2</td>
<td>Domain Geometries and Meshing</td>
<td>198</td>
</tr>
<tr>
<td>5.3</td>
<td>Boundary Conditions</td>
<td>200</td>
</tr>
<tr>
<td>5.4</td>
<td>Predictions of Contracting Nozzle Boundary Layer Development</td>
<td>202</td>
</tr>
<tr>
<td>5.5</td>
<td>Predictions of Nozzles with Parallel Extension</td>
<td>212</td>
</tr>
<tr>
<td>5.6</td>
<td>Closure</td>
<td>217</td>
</tr>
<tr>
<td>5.7</td>
<td>Figures</td>
<td>218</td>
</tr>
<tr>
<td>6</td>
<td>Conclusion and Further Work</td>
<td>251</td>
</tr>
<tr>
<td>6.1</td>
<td>Summary and Conclusions</td>
<td>251</td>
</tr>
<tr>
<td>6.2</td>
<td>Recommendations for Further Work</td>
<td>253</td>
</tr>
</tbody>
</table>
List of Figures

1.1 Rolls-Royce Trent 800 with Serrations .................................. 3
1.2 General Dynamics F16 Fighting Falcon with reheat .................... 5
1.3 Northrop-Grumman B2 Spirit .................................................. 5
1.4 Fairchild Republic A10 Thunderbolt .......................................... 6
1.5 Schematic of the Regions of a Free Jet ...................................... 10
1.6 Schematic Shock Cells in an Inviscid Underexpanded Jet Plume .......... 12
1.7 Nozzle Exit Boundary Layer Shape Factor Variation with Jet Reynolds Number .......................................................... 19
1.8 Turbulent Boundary Layer Profile Using Inner-Wall Scaling .......... 23
1.9 Variation of the Mean Velocity Profile During Relaminarisation .... 25
1.10 Variation of Boundary Layer Parameters During Relaminarization 27
1.11 The stages of relaminarisation .................................................... 27

2.1 Air Supply System ................................................................. 74
2.2 High Pressure Nozzle Test Facility- Nozzle Test Rig ................. 75
2.3 Schematic of the Nozzle Test Rig .............................................. 75
2.4 HPNTF Control Room ............................................................... 76
2.5 LU48 and LU60 Nozzle Geometries ............................................ 77
2.6 LU48(P) and LU60(P) Nozzle Geometries .................................. 77
2.7 Boundary Layer Pitot Probe Sensing Head .................................. 78
<table>
<thead>
<tr>
<th>FIGURE</th>
<th>DESCRIPTION</th>
</tr>
</thead>
<tbody>
<tr>
<td>2.8</td>
<td>Inlet Measurement Extension Duct</td>
</tr>
<tr>
<td>2.9</td>
<td>Nozzle Inlet Boundary Layer Traverse</td>
</tr>
<tr>
<td>2.10</td>
<td>Nozzle Inlet Total Pressure Variation (LU60, NPR 1.57)</td>
</tr>
<tr>
<td>2.11</td>
<td>Nozzle Exit Total Pressure Variation (LU60, NPR 1.58 )</td>
</tr>
<tr>
<td>2.12</td>
<td>Dual Probe for Nozzle Exit Measurements</td>
</tr>
<tr>
<td>2.13</td>
<td>Nozzle Exit Probe Traverse</td>
</tr>
<tr>
<td>2.14</td>
<td>Variation of Mean Velocity Profiles with Axial Distance</td>
</tr>
<tr>
<td>2.15</td>
<td>Variation of RMS Velocity Profiles with Axial Distance</td>
</tr>
<tr>
<td>2.16</td>
<td>LDA Liquid Droplet and Solid Particle Seeding Units</td>
</tr>
<tr>
<td>2.17</td>
<td>LDA Probe and Traverse</td>
</tr>
<tr>
<td>2.18</td>
<td>LDA Transmitter Probe Orientation</td>
</tr>
<tr>
<td>2.19</td>
<td>Interpolation of Pressure Profile to Determine Effective Wall Location (LU60, NPR ≈ 1.6)</td>
</tr>
<tr>
<td>2.20</td>
<td>Interpolation of Velocity Profile to Determine Effective Wall Location (LU60, NPR ≈ 1.6)</td>
</tr>
<tr>
<td>2.21</td>
<td>Determination of Skin Friction Using the Clauser Plot Method</td>
</tr>
<tr>
<td>2.22</td>
<td>Influence of Probe Incidence on Measured Pressure</td>
</tr>
<tr>
<td>2.23</td>
<td>Mean Velocity Profile With Error Bars for 99% Confidence Interval (LU60, NPR ≈ 1.68)</td>
</tr>
<tr>
<td>2.24</td>
<td>RMS Axial Velocity Profile With Error Bars for 99% Confidence Interval (LU60, NPR ≈ 1.68)</td>
</tr>
<tr>
<td>2.25</td>
<td>LDA Measurement Volume</td>
</tr>
<tr>
<td>3.1</td>
<td>Solid Model Representation of a Solution Domain</td>
</tr>
<tr>
<td>3.2</td>
<td>Test Case 1- Semi-logarithmic Mean Velocity Profiles</td>
</tr>
<tr>
<td>3.3</td>
<td>Test Case 1- Reynolds Shear Stress Profiles</td>
</tr>
<tr>
<td>3.4</td>
<td>Relaminarising Boundary Layer Experimental Geometries</td>
</tr>
</tbody>
</table>
LIST OF FIGURES

3.5 Test Case 2- Inlet Mean Axial Velocity Profiles ..................................... 114
3.6 Test Case 2- Semi-logarithmic Mean Velocity Profiles Profiles ..................... 115
3.7 Test Case 2- Inlet Turbulent Kinetic Energy Profiles ................................ 115
3.8 Test Case 2- Inlet Reynolds Shear Stress \( \mu' \nu' \) Profiles ......................... 116
3.9 Test Case 2- Contraction Region Mesh ..................................................... 116
3.10 Test Case 2- Boundary Layer Region Mesh ............................................... 117
3.11 Comparison of the Axial Development of the Reynolds Number, Shape Parameter, Skin-Friction Coefficient and the Acceleration Parameter for Test Case 2 .................................................. 118
3.12 Comparison of the Axial Development of the Non-dimensional Mean Axial Velocity for Test Case 2 .............................................................. 119
3.13 Comparison of the Axial Development of the Non-dimensional Reynolds Shear Stress \(-\mu' \nu'\) for Test Case 2 ................................................................. 120
4.1 Variation of LU48 Nozzle Inlet Boundary Layer Parameters with NPR ........ 150
4.2 Variation of LU60 Nozzle Inlet Boundary Layer Parameters with NPR ........ 151
4.3 Influence of Compressibility on Shape Factor \((LU60)\) .............................. 152
4.4 Influence of Compressibility on Momentum Thickness Reynolds Number \((LU60)\) 152
4.5 Correlation of Local Skin Friction Coefficient with Kinematic Momentum Thickness Reynolds Number \((LU48, LU60)\) .......................... 153
4.6 LU48 mean velocity NPR 1.3-1.8 (Inner Wall Scaling) ............................. 154
4.7 LU48 mean velocity NPR 1.8-2.4 (Inner Wall Scaling) ............................. 154
4.8 LU60 mean velocity NPR 1.3-1.8 (Inner Wall Scaling) ............................. 155
4.9 LU60 mean velocity NPR 1.8-2.4 (Inner Wall Scaling) ............................. 155
4.10 LU48 mean velocity NPR 1.3-1.8 (Outer Wall Scaling) ........................... 156
4.11 LU48 mean velocity NPR 1.8-2.4 (Outer Wall Scaling) ........................... 156
4.12 LU60 mean velocity NPR 1.3-1.8 (Outer Wall Scaling) ........................... 157
4.13 LU60 mean velocity NPR 1.8-2.4 (Outer Wall Scaling) ........................... 157
4.14 LU48 Nozzle Exit Boundary Layer Kinematic Momentum Thickness Reynolds Number Variation (Probe Data) ........................................ 158
4.15 LU48 Nozzle Exit Boundary Layer Kinematic Momentum Thickness Variation (Probe Data) ........................................ 158
4.16 LU48 Nozzle Exit Boundary Layer Kinematic Shape Factor Variation (Probe Data) ........................................ 159
4.17 LU48 Nozzle Exit Boundary Layer Profile Variation (NPR 1.3-1.6) (Probe Data) ........................................ 159
4.18 LU48 Nozzle Exit Boundary Layer Profile Variation (NPR 1.65-1.9) (Probe Data) ........................................ 160
4.19 LU48 Nozzle Exit Boundary Layer Clauser Plots (Probe Data) ........................................ 160
4.20 LU60 Nozzle Exit Boundary Layer Kinematic Momentum Thickness Reynolds Number Variation (Probe Data) ........................................ 161
4.21 LU60 Nozzle Exit Boundary Layer Kinematic Momentum Thickness Variation (Probe Data) ........................................ 161
4.22 LU60 Nozzle Exit Boundary Layer Kinematic Shape Factor Variation (Probe Data) ........................................ 162
4.23 LU60 Nozzle Exit Boundary Layer Profile Variation (NPR 1.3-1.6) (Probe Data) ........................................ 162
4.24 LU60 Nozzle Exit Boundary Layer Profile Variation (NPR 1.65-1.9) (Probe Data) ........................................ 163
4.25 LU60 Nozzle Exit Boundary Layer Clauser Plots (Probe Data) ........................................ 163
4.26 Influence of LDA Measurement Volume on Mean Axial Velocity ........................................ 164
4.27 Influence of LDA Measurement Volume on Clauser Plot Mean Axial Velocity ........................................ 164
4.28 Influence of LDA Measurement Volume on the RMS of Mean Axial Velocity ........................................ 165
4.29 Influence of Weighting on Mean Velocity ($LU60, NPR1.25$) ........................................ 166
4.30 Influence of Weighting on RMS Axial Velocity ($LU60, NPR1.25$) ........................................ 166
4.31 Influence of Weighting on Reynolds Shear Stress ($LU60, NPR1.25$) ........................................ 167
4.32 Influence of Weighting on Kinematic Shape Factor ($LU60$) ........................................ 167
<table>
<thead>
<tr>
<th>Figure</th>
<th>Description</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>4.33</td>
<td>Influence of Weighting on Kinematic Momentum Thickness Reynolds Number (LU60)</td>
<td>168</td>
</tr>
<tr>
<td>4.34</td>
<td>LU48 LDA Nozzle Exit Boundary Layer Kinematic Momentum Thickness Reynolds Number Variation</td>
<td>168</td>
</tr>
<tr>
<td>4.35</td>
<td>LU48 LDA Nozzle Exit Boundary Layer Kinematic Momentum Thickness Variation</td>
<td>169</td>
</tr>
<tr>
<td>4.36</td>
<td>LU48 LDA Nozzle Exit Boundary Layer Kinematic Shape Factor Variation</td>
<td>169</td>
</tr>
<tr>
<td>4.37</td>
<td>LU48 LDA Nozzle Exit Boundary Layer Profile Variation (NPR 1.3-1.75)</td>
<td>170</td>
</tr>
<tr>
<td>4.38</td>
<td>LU48 LDA Nozzle Exit Boundary Layer Profile Variation (NPR 1.8-2.4)</td>
<td>171</td>
</tr>
<tr>
<td>4.39</td>
<td>Comparison of Nozzle Inlet and Nozzle Exit Boundary Layer profiles (LU48)</td>
<td>171</td>
</tr>
<tr>
<td>4.40</td>
<td>LU48 LDA Nozzle Exit Boundary Layer RMS Axial Velocity Profile Variation (NPR 1.3-1.75)</td>
<td>172</td>
</tr>
<tr>
<td>4.41</td>
<td>LU48 LDA Nozzle Exit Boundary Layer Reynolds RMS Axial Velocity Profile Variation(NPR 1.8-2.40)</td>
<td>172</td>
</tr>
<tr>
<td>4.42</td>
<td>Variation of Peak RMS with NPR (LU48 and LU60)</td>
<td>173</td>
</tr>
<tr>
<td>4.43</td>
<td>LU60 LDA Nozzle Exit Boundary Layer RMS Axial Velocity Profile Variation (NPR 1.3-1.75)</td>
<td>174</td>
</tr>
<tr>
<td>4.44</td>
<td>LU60 LDA Nozzle Exit Boundary Layer RMS Axial Velocity Profile Variation (NPR 1.8-2.4)</td>
<td>174</td>
</tr>
<tr>
<td>4.45</td>
<td>LU48 LDA Nozzle Exit Boundary Layer Reynolds Shear Stress ($u'v'$) Profile Variation (NPR 1.3-1.75)</td>
<td>175</td>
</tr>
<tr>
<td>4.46</td>
<td>LU48 LDA Nozzle Exit Boundary Layer Reynolds Shear Stress ($u'v'$) Profile Variation(NPR 1.8-2.40)</td>
<td>175</td>
</tr>
<tr>
<td>4.47</td>
<td>Variation of Peak Reynolds Shear Stress ($u'v'$) with NPR (LU48 and LU60)</td>
<td>176</td>
</tr>
<tr>
<td>4.48</td>
<td>LU60 LDA Nozzle Exit Boundary Reynolds Shear Stress ($u'v'$) Profile Variation (NPR 1.3-1.75)</td>
<td>177</td>
</tr>
<tr>
<td>4.49</td>
<td>LU60 LDA Nozzle Exit Boundary Reynolds Shear Stress ($u'v'$) Profile Variation (NPR 1.8-2.4)</td>
<td>177</td>
</tr>
<tr>
<td>4.50</td>
<td>Location of LDA Traverse Line Relative to Nozzle Lip Expansion Fan (LU60, NPR = 2.40)</td>
<td>178</td>
</tr>
</tbody>
</table>
4.51 Influence of Prandtl-Meyer Expansion Fan on Measured Boundary Layer Velocity Profile (LU60(P)) .............................................. 179
4.52 LU48(P) Nozzle Exit Boundary Layer Profiles ........................................ 179
4.53 LU60(P) Nozzle Exit Boundary Layer Profiles ........................................ 180
4.54 LU48(P) Clauser Plots ................................................................. 180
4.55 LU60(P) Clauser Plots ................................................................. 181
4.56 LU48(P) Nozzle Exit Boundary Layer RMS Axial Velocity Profiles ........ 181
4.57 LU60(P) Nozzle Exit Boundary Layer RMS Axial Velocity Profiles ........ 182
4.58 LU48(P) Nozzle Exit Boundary Layer Reynolds Shear Stress ($u'v'$) Profiles 182
4.59 LU60(P) Nozzle Exit Boundary Layer Reynolds Shear Stress ($u'v'$) Profiles 183
4.60 Comparison of Axial Variation of Mean and RMS Axial Velocity Profiles (NPR = 1.50) .............................................. 184
4.61 Comparison of Axial Variation of Mean and RMS Axial Velocity Profiles (NPR = 1.88) .............................................. 185
4.62 Profiles of Axial Velocity (NPR = 1.50) ........................................ 186
4.63 Profiles of Axial RMS Velocity (NPR = 1.50) .................................... 187
4.64 Profiles of Shear Stress $u'v'$ (NPR = 1.50) ....................................... 188
4.65 Jet Plume Comparison (NPR=1.50) ............................................... 189
4.66 Jet Plume Comparison (NPR=1.88) ............................................... 190
4.67 Growth of Mixing Layer ............................................................... 190
4.68 Comparison of Axial Variation of Mean and RMS Axial Velocity Profiles (NPR = 2.40) .............................................. 191
4.69 Axial Variation of $V_{rms}$ (NPR = 2.40) ........................................ 192
4.70 Schlieren Image of Underexpanded Jet Plumes .................................. 193
4.71 Profiles of Axial Velocity (NPR = 2.40) ........................................ 194
4.72 Profiles of Axial RMS Velocity (NPR = 2.40) .................................... 195
4.73 Profiles of Shear Stress $u'v'$ (NPR = 2.40) ....................................... 196
5.1 LU60 Nozzle Geometry ........................................... 219
5.2 LU60 Nozzle With Parallel-Walled Extension Geometry .............. 219
5.3 LU60 Nozzle With Jet Plume Region Geometry .......................... 220
5.4 LU60 Mesh .......................................................... 220
5.5 LU60 Nozzle With Parallel-Walled Extension Mesh ..................... 221
5.6 LU60 Nozzle With Jet Plume Mesh ................................. 221
5.7 Inlet Profiles for $NPR = 1.50$ ...................................... 222
5.8 Variation of Boundary Layer Parameters with Axial Location for $NPR=1.88$ 223
5.9 Contours of Static Pressure for Prediction without Jet Plume ($NPR=1.88$) 224
5.10 Contours of Static Pressure for Prediction with Jet Plume ($NPR=1.88$) ... 224
5.11 Contours of Axial Velocity for Prediction without Jet Plume ($NPR=1.88$) ... 225
5.12 Contours of Axial Velocity for Prediction with Jet Plume ($NPR=1.88$) ... 225
5.13 Comparison of Mean Velocity Profiles .................................. 226
5.14 Comparison of Reynolds Shear Stress $(\overline{\rho u'v'})$ Profiles ......................... 226
5.15 Comparison of Turbulent Kinetic Energy Profiles ......................... 227
5.16 Variation of Boundary Layer Parameters with Axial Location for $NPR=1.50$ 228
5.17 Variation of Boundary Layer Parameters with Axial Location for $NPR=2.40$ 229
5.18 Variation of Mean Velocity Profiles for $NPR = 1.88$ ................. 230
5.19 Variation of Static Pressure Gradient Profiles for $NPR = 1.88$ ........ 231
5.20 Variation of Reynolds Shear Stress $\overline{\rho u'v''}$ Profiles for $NPR = 1.88$ .... 231
5.21 Variation of Turbulent Kinetic Energy Profiles for $NPR = 1.88$ .......... 232
5.22 Comparison of Mean Velocity Profiles for Varying $NPR$ ............... 233
5.23 Comparison of Reynolds Shear Stress $\overline{\rho u'v''}$ Profiles for Varying $NPR$ ..... 234
5.24 Comparison of Turbulent Kinetic Profiles for Varying $NPR$ ............. 235
5.25 Comparison of Internal and External Predicted Velocity Profiles ($NPR = 1.88$) 236
5.26 Comparison of Internal and External Predicted Velocity Profiles (Clauser Plot) \((NPR = 1.88)\) ........................................... 236

5.27 Comparison of Experimental and Predicted Mean Velocity Profiles \((NPR = 1.50)\) .................................................. 237

5.28 Comparison of Experimental and Predicted Mean Velocity Profiles \((NPR = 1.88)\) .................................................. 237

5.29 Comparison of Experimental and Predicted Mean Velocity Profiles \((NPR = 2.40)\) .................................................. 238

5.30 Comparison of Measured and Predicted Nozzle Exit Turbulent Kinetic Energy Profiles ........................................... 239

5.31 Comparison of Measured and Predicted Nozzle Exit Reynolds Shear Stress Profiles .................................................. 240

5.32 Variation of Boundary Layer Parameters Within Parallel Nozzle Extension for \(NPR=1.50\) .............................................. 241

5.33 Variation of Boundary Layer Parameters Within Parallel Nozzle Extension for \(NPR=1.88\) .............................................. 241

5.34 Variation of Boundary Layer Parameters Within Parallel Nozzle Extension for \(NPR=2.40\) .............................................. 242

5.35 Recovery of near-wall Region Mean Velocity Profile for \(NPR=1.50\) .................................................. 242

5.36 Recovery of Outer Region of the Mean Velocity Profile for \(NPR=1.50\) .................................................. 243

5.37 Recovery of the Reynolds Shear Stress \(\overline{\rho u' v'}\) Profile for \(NPR=1.50\) .................................................. 243

5.38 Recovery of near-wall Region Mean Velocity Profile for \(NPR=2.40\) .................................................. 244

5.39 Recovery of Outer Region of the Mean Velocity Profile for \(NPR=2.40\) .................................................. 244

5.40 Recovery of the Reynolds Shear Stress \(\overline{\rho u' v'}\) Profile for \(NPR=2.40\) .................................................. 245

5.41 Comparison of Predicted and Measured Exit Boundary Layers for Nozzles with Parallel Extensions .................................................. 245

5.42 Comparison of Predicted and Measured Turbulent Kinetic Energy Profiles \((NPR = 1.50)\) .................................................. 246

5.43 Comparison of Predicted and Measured Reynolds Shear Stress Profiles \((NPR = 1.50)\) .................................................. 246
<table>
<thead>
<tr>
<th>Figure</th>
<th>Description</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>5.44</td>
<td>Contours of Static Pressure in the Nozzle Parallel Extension for NPR=1.50</td>
<td>247</td>
</tr>
<tr>
<td>5.45</td>
<td>Contours of Axial Velocity Near the Wall in the Parallel Extension for NPR=1.50247</td>
<td>247</td>
</tr>
<tr>
<td>5.46</td>
<td>Contours of Static Pressure in the Nozzle and Parallel Extension for NPR=1.88248</td>
<td>248</td>
</tr>
<tr>
<td>5.47</td>
<td>Contours of Mach Number Near the Wall in the Parallel Extension for NPR=1.88248</td>
<td>248</td>
</tr>
<tr>
<td>5.48</td>
<td>Contours of Static Pressure in the Nozzle and Parallel Extension for NPR=2.4</td>
<td>249</td>
</tr>
<tr>
<td>5.49</td>
<td>Contours of Mach Number in the Nozzle and Parallel Extension for NPR=2.4</td>
<td>249</td>
</tr>
<tr>
<td>5.50</td>
<td>Parallel Extension Centreline Static Pressure Variations</td>
<td>250</td>
</tr>
<tr>
<td>6.1</td>
<td>Predicted Variation of the Acceleration Parameter with Total Temperature</td>
<td>255</td>
</tr>
<tr>
<td></td>
<td><em>(LU60, NPR = 1.88)</em></td>
<td></td>
</tr>
</tbody>
</table>
List of Tables

2.1 Measurement Volume Characteristics ........................................... 51
2.2 Measurement Volume Characteristics With Beam Expanders ............ 52
2.3 Key to Figures 2.2 and 2.3 ......................................................... 76
3.1 $k-\varepsilon$ Turbulence Model Coefficients ................................... 95
3.2 Transport Equation Diffusion Coefficients and Source Terms ............ 98
3.3 Case 2 Experimental Conditions ................................................ 105
3.4 Comparison of computational and experimental inlet conditions ......... 106
3.5 Comparison of Key Boundary Layer Parameters (Warnack, Case 2) .... 108
4.1 Measurement Volume Characteristics With Beam Expanders and 38mm separation ................................................................. 132
4.2 Boundary Layer Parameters for Nozzle With Parallel Walled Extension ... 138
5.1 Comparison of Measured and Predicted Inlet Boundary Layer Parameters ... 202
5.2 Comparison of Boundary Layer Parameters from Internal and External Measurement Planes .......................................................... 210
5.3 Comparison of Measured and Predicted Nozzle Exit Boundary Layer Parameters for The Contracting Nozzle ........................................ 211
5.4 Comparison of Measured and Predicted Boundary Layer Parameters for Nozzle With Parallel Wall Extension ..................................... 215
Nomenclature

Latin Characters

\( a \)  
- speed of sound
\( a_1 \)  
- structure parameter
\( B \)  
- constant in logarithmic law of the wall
  - shear layer width
\( c \)  
- confidence level coefficient
\( c_{11} \)  
- constant in optical transformation matrix
\( c_{12} \)  
- constant in optical transformation matrix
\( c_{21} \)  
- constant in optical transformation matrix
\( c_{22} \)  
- constant in optical transformation matrix
\( c_f \)  
- local skin friction coefficient
\( C_p \)  
- specific heat at a constant pressure
\( C_v \)  
- specific heat at a constant volume
\( D \)  
- nozzle exit diameter
  - pipe diameter
\( D_{ij} \)  
- deformation tensor
\( E \)  
- beam expansion ratio
  - fluid specific total energy
\( f_1 \)  
- damping function used in Launder-Sharma turbulence model
\( f_2 \)  
- damping function used in Launder-Sharma turbulence model
\( f_4 \)  
- Doppler frequency
\( f_s \)  
- sampling frequency
\( f_\mu \)  
- damping function used in Launder-Sharma turbulence model
\( H \)  
- fluid specific total enthalpy
  - Pitot probe sensing tip height
Nomenclature

$H_{12}$ boundary layer shape factor
$I_1$ integral term in definition of axisymmetric displacement thickness
$I_2$ integral term in definition of axisymmetric momentum thickness
$k$ turbulent kinetic energy
$K$ acceleration parameter
$K_c$ compressible acceleration parameter
$\ell$ dissipation length
$l_t$ turbulent length scale
$L$ characteristic length
$m$ number of degrees of freedom in normal distribution
$M$ Mach number
$M_c$ convective Mach number
$M_j$ fully expanded jet Mach number
$n$ number of samples for ensemble average
$p$ static pressure
$P$ total pressure
$P_k$ turbulent kinetic energy production term
$q$ heat flux
$R$ universal gas constant $R = c_p - c_v$
$R$ pipe radius
$r$ radius
$Re$ Reynolds number
$Re_D$ Pipe Reynolds number
$Re_j$ Jet Reynolds number
$Re_t$ turbulence Reynolds number
$Re_{unit}$ unit Reynolds number
$Re_\theta$ momentum thickness Reynolds number
$S$ standard deviation
$S_\phi$ source or sink of a scalar variable
$t$ static temperature
$T$ total temperature
$T_j$ jet total temperature
$T_o$ ambient total temperature
$u^+$ non-dimensional velocity
$u_r$ friction velocity
$U$ Cartesian velocity component
$U_{BSA}$ LDA measured instantaneous velocity
$U_j$ jet velocity
Nomenclature

- $v_n$: normal velocity component
- $V$: Cartesian velocity component
- $W$: Cartesian velocity component
- $x_o$: jet virtual origin
- $x_{pc}$: potential core length
- $y$: wall normal distance
- $y^+$: non-dimensional wall normal distance
- $y_c$: wall normal distance to centre of measurement volume or probe

Greek Characters

- $\alpha$: non-dimensional velocity gradient
- $\delta$: boundary layer thickness
- $\delta_{ij}$: free shear layer thickness
- $\delta^*$: probe displacement
- $\delta_{ij}$: Krönecker delta function, $\delta_{ij} = 1$ if $i = j$, $\delta_{ij} = 0$ if $i \neq j$
- $\delta^*$: displacement thickness
- $\Delta$: Rotta-Clauser length
- $\Delta_{p}$: non-dimensional pressure gradient parameter
- $\Delta_{ts}$: inter-sample time
- $\epsilon$: confidence of a sampled normally distributed random variable
- $\epsilon$: turbulent dissipation rate
- $\tilde{\epsilon}$: isotropic turbulent dissipation rate
- $\phi$: scalar variable
- $\gamma$: intermittency ratio
- $\eta_i$: ratio of specific heats
- $\eta_i$: LDA weighting factor
- $\rho$: density
- $\kappa$: von Kármán constant
- $\mu$: absolute molecular viscosity
- $\mu_t$: turbulent viscosity
- $\sigma$: variance
- $\sigma_k$: turbulent Prandtl number for kinetic energy
- $\sigma_t$: turbulent Prandtl number
- $\sigma_e$: turbulent Prandtl number for dissipation rate
- $\tau$: shear stress
Nomenclature

$\tau_{ij}$  shear stress tensor
$\nu$  kinematic molecular viscosity
$\nu_t$  kinematic turbulent viscosity

Operators

$\bar{\cdot}$  Reynolds averaged quantities
$\bar{\cdot}'$  Favre averaged quantities

Subscripts

CL  evaluated at centreline
i  kinematic form
  Cartesian direction index
j  Cartesian direction index
k  Cartesian direction index
m  measured
pc  value in the potential core
w  evaluated at wall
o  evaluated at nozzle exit
$\delta$  evaluated at boundary layer edge
$\infty$  evaluated at free stream
0.05  evaluated at $0.05\times$ centreline state
0.10  evaluated at $0.10\times$ centreline state
0.95  evaluated at $0.95\times$ centreline state

Acronyms

BSA  Burst Spectrum Analyser
CFD  Computational Fluid Dynamics
CFL  Courant-Friedrichs-Lewy
DFT  Discrete Fourier Transform
<table>
<thead>
<tr>
<th>Abbreviation</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>DNS</td>
<td>Direct Numerical Simulation</td>
</tr>
<tr>
<td>DSP</td>
<td>Digital Signal Processor</td>
</tr>
<tr>
<td>HPNTF</td>
<td>High Pressure Nozzle Test Facility</td>
</tr>
<tr>
<td>ICAO</td>
<td>International Civil Aviation Organisation</td>
</tr>
<tr>
<td>IR</td>
<td>Infra-red</td>
</tr>
<tr>
<td>LDA</td>
<td>Laser Doppler Anemometer</td>
</tr>
<tr>
<td>LES</td>
<td>Large Eddy Simulation</td>
</tr>
<tr>
<td>LRN</td>
<td>Low Reynolds Number</td>
</tr>
<tr>
<td>NPR</td>
<td>Nozzle Pressure Ratio</td>
</tr>
<tr>
<td>NASA</td>
<td>National Aeronautics and Space Administration</td>
</tr>
<tr>
<td>QUICK</td>
<td>Quadratic Upwind Interpolation for Convective Kinetics</td>
</tr>
<tr>
<td>RANS</td>
<td>Reynolds-Averaged Navier Stokes</td>
</tr>
<tr>
<td>STOVL</td>
<td>Short Take-off and Vertical Landing</td>
</tr>
</tbody>
</table>
Chapter 1

Introduction

1.1 Introduction

The principle of jet reaction may have been first demonstrated by the Greek philosopher and mathematician Hero of Alexandria around the first century BC with his steam powered Aeolipile[1], but it was not until the pioneering work of Von Ohain and Whittle, some 2000 years later, that the principle would revolutionise aircraft propulsion in the form of the gas turbine engine. Ever since the first flights of aircraft powered by the gas turbine engine in August 1939 (Von Ohain) and May 1941 (Whittle)[2] engineers have shown considerable interest in the development and mixing of the high speed jet plumes which are formed downstream of the exhaust nozzles of gas turbine engines. The exhaust nozzle and resulting jet flow form the main engineering application motivating the research discussed in this thesis and, hence, it is appropriate to review briefly the reasons why exhaust nozzles and jet flows are of such engineering significance. There are several specific technical reasons for this which can be broadly divided into two application areas, civil aviation and military aviation.

1.1.1 Exhaust Nozzle Applications-Civil Aviation

Over the last three decades, civil aviation has seen a five-fold increase in volume, including a doubling in the amount of air freight since 1990. Currently more than one half of the UK population makes at least one flight each year. A recent white paper by the Department for Transport [3] revealed that the expected growth in the popularity of air travel, fueled largely by decreasing costs and greater competition from budget ‘no frills’ carriers, is expected to produce a two to three-fold increase in demand over current levels over the next three decades.
1.1 Introduction

This increase in air travel also brings about an increase in the environmental impact of the operating aircraft fleet. One area of concern is noise, particularly the impact that it has on populations in the vicinity of airports. Although aircraft noise levels have typically reduced by 75% since the first generation of civil jet aircraft of the 1950s and 1960s [3], the rapid increase in the volume of air traffic is at risk of cancelling out the benefits of aircraft noise reduction brought about through technological advances. As a result, the UK government aims to reduce this impact through more stringent legislation, economic measures (including differential landing costs based on aircraft noise levels at airports where noise problems exist) and promoting quiet aircraft technologies through funding research.

The International Civil Aviation Organisation (ICAO), who are responsible for aircraft noise regulations, have focused on reducing aircraft noise at source through the tightening of noise certification standards for new aircraft [4]. The next stage of noise regulations (ICAO Annex 16, Chapter 4, to be introduced in 2006) will see a 10EPNdB reduction on previous requirements. This increasingly stringent legislation is constantly forcing engineers to discover and develop new ways of reducing aircraft noise. One of the largest sources of aircraft noise (particularly during take-off where engine thrust requirements are at their greatest and aircraft are in closest proximity to densely populated areas) is the noise produced by the engine exhaust jet plume. The additional noise reduction required by new legislation needs to be met through new means. As a result, research efforts have been directed at influencing the way in which the near-field region of the jet plume develops either through active (such as acoustic forcing) [5] or passive means (such as tabs [6] and crown-like structures [7]), manipulating the development of the initial jet/ambient shear layer. Through careful control of the way in which the jet plume develops it is possible to reduce noise production, particularly at the low frequencies which are of greatest concern to the designer as these travel much greater distances than high frequency noise. The effectiveness of passive mixing devices in reducing perceived noise levels but without introducing significant thrust penalties has been demonstrated in a commercial context by nozzle trailing edge treatments known as chevrons or serrations. Full scale tests of a Rolls-Royce Trent 800 with serrations, mounted on a Boeing 777 (shown in Figure 1.1) achieved a 4EPNdB noise reduction [8] (where EPNdB refers to a sound pressure level scale, weighted to reflect the sensitivity of human hearing across the frequency spectrum) and nozzles fitted with chevrons on the core nozzle of a CFM56 engine achieved a reduction of 2.6EPNdB [9]. Laboratory tests of similar chevroned nozzles show that these reductions in noise can be achieved with a thrust penalty as low as 0.06% [10].

Concerns surrounding exposure to jet noise are not limited to those outside of the aircraft. New issues are being raised about the health and safety of those exposed to noise within the cockpit and cabin environments. As aircraft take-off mass increases, thrust re-
1.1 Introduction

Requirements also increase. The required thrust is being achieved by raising nozzle pressure ratios (NPR, defined as the ratio of the total pressure at inlet to the nozzle to the ambient static pressure), such that future engine nozzles may operate at NPRs above the point at which the nozzle chokes during cruise (NPRs of up to 2.6 are used on the Rolls-Royce Trent family of engines). The resulting formation of shock-cells in the jet plume produces shock noise, which, depending on the location and mounting of the engines, may result in noise penetrating into the cockpit and cabin areas. Research is focusing on developing a better understanding of such noise propagation [11] and controlling shock noise at source [12]. It is clear from this brief discussion that within the civil aviation sector interest will continue to focus on improved understanding of jet plume flow development near the nozzle exit.

Figure 1.1: Rolls-Royce Trent 800 with Serrations

1.1.2 Exhaust Nozzle Applications-Military Aviation

Currently, the main driving force behind research into jet plumes originating from combat aircraft is the improvement of aircraft engine stealth. Infra-red (IR) radiation, emitted by any heat sources around the aircraft, is often used as a means of passive aircraft detection by other aircraft and anti-aircraft munitions [13], [14]. The largest source of IR radiation is of course the aircraft’s engines, in particular the engine afterburners and exhaust nozzles where temperatures are at their greatest. Although it is not possible to remove an aircraft’s
IR signature completely, through careful design it is possible to reduce it, therefore reducing the likelihood of detection.

In terms of the location of aircraft engines, using unconventional configurations that move away from the familiar circular exhaust nozzle configuration located at the rear of the aircraft (as illustrated by the General Dynamics F16 Fighting Falcon in Figure 1.2) can reduce an aircraft’s IR signature by ensuring hot parts of the engine cannot be seen from below. Such configurations bury the engines within the structure (such as the Northrop-Grumman B-2 Spirit, Figure 1.3) or shield the engines using parts of the airframe to reduce IR observability (Fairchild Republic A-10 Thunderbolt, Figure 1.4). These configurations need careful investigation if the interaction between the jet plume and surrounding structures is to be fully understood at the design stage to ensure that the jet flow does not affect any control systems that may be in close proximity, or indeed cause excessive heating of the airframe.

Another major source of IR radiation is the high temperature jet plume itself, particularly if the aircraft is in reheat and operating at off-design conditions (as shown in Figure 1.2) where static temperatures within the shock cells are above ideally expanded conditions [15]. In order to reduce IR observability of the jet plumes considerable effort has been invested into mixing enhancement devices in order to reduce jet fluid temperatures rapidly. Mixing devices such as mechanical tabs, which protrude into the jet at the nozzle exit, enhance mixing rates through the formation of streamwise vortical structures. Unconventional nozzle geometries such as rectangular [6] and elliptical nozzles [16] have also been shown to increase mixing through distortion of vortical structures which increase the surface area of the jet plume.

Modern stealth technologies also include methods to reduce the aircraft’s radar signature. One such method is the signature shaping technique which include the careful shaping of leading and trailing edges and ensuring that adjacent surfaces are not at 90° to each other [17]. Such design parameters may also need to be applied to the design of the exhaust nozzles as illustrated by the nozzles of the Northrop-Grumman B-2 Spirit, Figure 1.3. Incorporating such methods of reducing both radar and infra-red observability may result in the reduction of nozzle performance and, therefore, a trade-off between observability and performance is required during the design stage.

In order to resolve issues such as IR emission, it is obvious that a better understanding of the flowfield mechanisms responsible for jet development and mixing is desirable, particularly when considering the systematic optimisation of mixing devices. Several methods of investigation are available to the engineer, ranging from numerical simulations to small-scale and full-scale experimental investigations which are therefore now further discussed.
1.1 Introduction

Figure 1.2: General Dynamics F16 Fighting Falcon with reheat

Figure 1.3: Northrop-Grumman B2 Spirit
1.2 Experimental Studies of Nozzle Performance and Jet Plume Aerodynamics at Reduced Scale

Whether the design driver is related to civil or military applications, several options are open to the engineer when studying engine exhaust jet plumes in order to develop a better understanding of the physical mechanisms that are responsible for jet mixing and noise or to develop novel methods of controlling jet plume development to meet new legislation or low-observability requirements. Such options are flight testing, full-scale wind tunnel testing, full-scale static testing, numerical simulations using Computational Fluid Dynamics (CFD) and laboratory-based, small-scale model testing.

Flight testing provides a means of gathering data in a realistic flight environment and at speeds in the transonic and supersonic flight regimes which are often difficult to recreate in wind tunnel facilities but are encountered during the everyday operation of the devices. One example of flight testing is the noise measurements of the Rolls-Royce Trent 800 with serrated nozzles to reduce jet noise described in Nesbitt et al. [18] and shown in Figure 1.1 and the work presented by Orme et al. [19] of flight tests conducted to investigate thrust vectoring. However, flight testing brings with it numerous disadvantages. In order to test new propulsion systems, a suitable test bed is required, either by designing a new vehicle or adapting an existing one (such as the modified Trent 800 engine used to test serrated nozzles, or the modified F15 used to test vectoring nozzles). The modification and operation
1.2 Experimental Studies of Nozzle Performance and Jet Plume Aerodynamics at Reduced Scale

of such flying test beds is extremely expensive. The data gathered are also usually limited to proving the new device (for example, engine performance and noise reduction) as inflight measurements of the jet plume is not practical. As a result, flight testing is usually reserved for the last phase of the development of new devices where the new technologies have been developed through alternative means such as small scale testing and numerical predictions.

Full-scale wind tunnel tests are another option available to the engineer, where realistic flight Reynolds numbers can be achieved in a controlled environment, and are conducive to detailed measurement. Such full scale wind tunnel tests are presented in the work of Smith et al. [20] on a powered full-scale model of a STOVL aircraft to study propulsion-induced aerodynamic effects when operating in ground effect. Unfortunately, due to their limited numbers, large sizes, operating overheads and low productivity full-scale wind tunnel facilities (such as the 120x80ft tunnel at NASA’s Ames Research Centre) are generally not feasible for use in commercial projects, and are instead limited to military projects [21] and subsonic speeds.

An alternative experimental method for full scale testing of propulsion systems is static testing, whereby the jet issues into a quiescent environment; one example is the full-scale investigation of Mabe et al [22] into exhaust nozzles with variable geometry chevrons. This offers cost advantages over flight and wind tunnel testing as a flight stream is not used, but the lack of a flight stream does not allow its influence on the development of the jet plume at different flight conditions to be investigated. Like full-scale wind tunnel and flight testing, full-scale static tests are expensive and usually reserved for the end of the design process.

With computational speed doubling every 18 months, numerical methods are playing an ever greater role in the investigation of jet plume development, aeroacoustics and the development and refinement of mixing devices at small and full-scale. Numerical methods also allow the simulation of forward flight, which may otherwise be difficult to obtain in an experimental facility. Studies into novel mixing devices using numerical methods include the work presented by Kenzakowski et al. [23], Thomas et al. [24] and Birch et al. [25] on nozzles with tabs and chevrons, and distributed nozzles by Gaeta et al. [26]. The use of numerical methods allows various configurations to be tested across a wide range of operating conditions relatively inexpensively, providing a useful design guide and avoiding commitment to expensive experimental testing early on in the design process. However, despite the obvious cost advantages of numerical methods over experimental methods they are not without their limitations. Birch et al.[25] and Nesbitt et al. [18] stated that current Reynolds Averaged Navier Stokes (RANS) models are not capable of predicting mixing well, even in simple axisymmetric jet flows, and pointed out the lack of understanding of the modeling requirements for predicting complicated jet flows, such as those experienced when chevrons
are employed. Numerical methods such as Direct Numerical Simulation (DNS) and Large Eddy Simulation (LES) are computationally expensive and DNS is limited to low Reynolds number flows due to the need to resolve the smallest turbulent structures. In addition, the development and improvement of computational methods are still dependent on experimental data for validation.

Hence, reduced or small-scale model testing in a laboratory offers a compromise between the advantages of experimental testing and the large costs and experimental difficulties that are attributable to testing at large and full scales. Test can be performed in ‘academic’ facilities and as overheads, turn-around times and model production cost are much lower, small scale testing allows experimental testing to be part of the design process from the outset. Examples of using small-scale testing to investigate jet mixing enhancement and control were presented by Saiyed et al. [10], Krothapalli et al. [6], Doty et al. [27] (who used small-scale testing to investigate the influence of nozzle trailing edge modifications on jet mixing) and the work presented by Lawson et al. [28] (who describe the use of small scale testing to investigate the problems associated with impinging jets).

Although there are clear advantages to using small-scale testing of nozzle designs and associated jet flows to serve as a design aid for full-scale designs, it is important to understand the circumstances under which small-scale model results may be representative (or not) when compared to full-scale. Many factors influence the development of the initial region of a jet, ranging from operating conditions such as jet temperature and nozzle pressure ratio (or jet Mach number) (which are independent of nozzle scale), but also jet Reynolds number (a non-dimensional parameter which relates the relative significance of viscous effects to fluid inertia effects, defined as $Re = \frac{\rho U L}{\mu}$, where L is a characteristic length of the flow, typically the nozzle exit diameter), and finally the nozzle exit boundary layer; both of the latter parameters may be highly influenced by varying nozzle scale. The significance of these parameters is therefore discussed in detail in the following sections.

1.3 The Influence of Nozzle Exit Conditions on Jet Shear Layer Development

Before discussing factors that influence the development of the turbulent jet plume, it is prudent to describe briefly the formation of free jets and their main regions. The free jet is an example of a free shear layer flow, since it develops in the absence of solid boundaries [29]. The jet shear layers form as a result of high speed fluid issuing from the nozzle exit into a region that is either at rest (quiescent) or coflowing. An example of the near-field of a developing free jet for a sub-critical NPR (subsonic flow) is illustrated in Figure 1.5.
1.3 The Influence of Nozzle Exit Conditions on Jet Shear Layer Development

As the fluid leaves the nozzle, two distinct regions form, a potential core of fluid at the jet velocity (usually non-turbulent) and an annular (usually turbulent) shear layer which surrounds the potential core and forms due to the difference in velocity between jet and ambient fluid. When the jet issues from a nozzle the thickness and turbulence properties of the initial region of the shear layer are to some extent dependent on the boundary layer which has developed upstream of the nozzle exit. As the jet moves further downstream, the width of the shear layer grows as a result of entrainment of ambient fluid into the jet plume by the turbulent structures within the shear layer. The volume of the flow contained within the potential core decreases due to mixing until the point at which the inner edge of the annular shear layer extends to the centreline of the jet and the potential core closes. At this point the jet plume is fully turbulent [30]. The turbulent jet continues to entrain ambient fluid beyond the end of the potential core and, after a transition zone, reaches a self-similar region. As the total momentum flow rate of the jet is constant (no ambient static pressure gradients at sub-critical NPRs), the addition of the entrained fluid results in a continual decrease in the velocity along the axis of the jet. Included in Figure 1.5 is the virtual origin, the location of a hypothetical point source of momentum for the jet, based on the intercept of extrapolated curves of the spread of the outer jet edge from the self-similar region. Shifts in the virtual origin (for different nozzles for example) is often used in the literature reviewed below as a means of capturing variations in the spreading of the jet in the initial and transition region due to nozzle-specific influences. At large axial distances downstream of the nozzle exit (typically greater than about 30 times the nozzle exit diameter [29]) the flow behaviour (spreading rate, centreline decay rate) becomes independent of the initial conditions and profiles of mean flow properties collapse onto a single curve when non-dimensionalised using suitable scaling (self-similarity, Hinze [31] and Townsend [32]). The same cannot be said of the initial region of the jet where the development of the jet is dependent on nozzle exit conditions. Papadopoulos and Pitts [33] have shown that conditions at the nozzle exit are important in the development of the near-field region of the jet, in particular the potential core, the initial shear layers and the transition region. In terms of the engineering applications of exhaust nozzle flows and jet flows described above (both civil and military), it is this initial region that is of primary interest since the first 20 or so diameters of development is much more influential in establishing the level of noise production and jet IR signature than the self-similar region of the jet plume. Hence, it is important to review those factors which are influential in the development of this near-field region, since these are the phenomena which must be representative between full-scale and small-scale tests if these are to be confidently used in design.

There are several factors that influence the near-field development of the jet shear layer and transition region: nozzle NPR or Mach number (since most practical aeronautical ex-
haust jet flows involve speeds where the jet velocity is close to or exceeds the speed of sound) and jet initial temperature and density (particularly in the case of jet plumes developing downstream of military exhaust nozzles), finally, as noted above jet Reynolds number and the state and thickness of the nozzle exit boundary layers are also observed to exert an influence on near-field flow development. The following sections discuss these in more detail.

Figure 1.5: Schematic of the Regions of a Free Jet
1.3 The Influence of Nozzle Exit Conditions on Jet Shear Layer Development

1.3.1 The Influence of Jet Exit NPR or Mach Number, Jet Density and Jet Temperature

In practical aeronautical jet flows, nozzle exit temperatures are often high and in addition jet exit velocities correspond to Mach numbers which are high subsonic or supersonic; this implies that variable density and compressibility effects must be considered. The jet exit Mach number \( M_j \) is determined by the Nozzle Pressure Ratio (NPR) and the nozzle configuration itself.

Until now, the description of the structure of nozzle-generated jet flows has been limited to subsonic jet plumes. Aircraft exhaust nozzles are often operated at high pressures such that the NPR is above the critical value (1.893 for air). At larger NPRs a convergent nozzle will choke and the jet plume enters the underexpanded regime. For these conditions the flow has not ideally expanded within the nozzle and static pressure at the nozzle exit is greater than ambient. The mismatch in pressure means the flow must expand (accelerate) in order to reach equilibrium with the surrounding ambient fluid, which results in the formation of inviscid, complex shock cells as illustrated in Figure 1.6. At this point a brief description of these inviscid structures is prudent.

Starting at the nozzle lip, reduction of the static pressure to ambient is achieved through the formation of a Prandtl-Meyer expansion fan emanating from the nozzle lip which reduces the static pressure to an ambient level at the jet boundary. Toward the centreline, the static pressure is reduced to values below ambient (denoted by '-' in Figure 1.6) due to the coalescence of expansion fans from all round the nozzle exit. As the expansion fan propagates outwards toward the edge of the jet it encounters a constant pressure boundary at the outer edge of the jet [34]. In order to maintain a constant pressure at this boundary the expansion waves are reflected as compression waves which coalesce to form an oblique shock wave. Towards the centerline, the coalescence of the compression fans from the different sides of the jet results in a region of flow with a pressure greater than ambient (denoted by '+' in Figure 1.6). This process of expansion and compression continues, forming quasiperiodic structures referred to as shock-cells in the supersonic region of the jet plume, until equilibrium is reached. Further details on the underexpanded flow regime can be found in Anderson [34]. In the shock-containing region of the jet the flow properties \( t, P, p, \rho \), local Mach number and velocities all vary as the flow accelerates and decelerates. Note that the reflection process at the jet boundary will only be ideal (perfect) from a sharply-defined boundary edge. As the shear layer grows with distance, the reflection process will become more and more diffusive and the strength of the over/under pressures relative to ambient in the shock cells decreases with axial distance.
1.3 The Influence of Nozzle Exit Conditions on Jet Shear Layer Development

Figure 1.6: Schematic Shock Cells in an Inviscid Underexpanded Jet Plume

Let us now return to the influence of density, temperature and compressibility on jet near-field shear layer development. It is useful to separate the effect on shear layer development of jet/ambient fluid density and velocity differences under incompressible conditions from velocity and density effects induced by compressibility influences.

In order to decouple the effects of jet Mach number (compressibility effects) arising from density and velocity ratio across the jet free shear layer from the incompressible influence of velocity and density ratio, it is necessary first to record the spreading rate of an incompressible shear layer as velocity and density ratio across the shear layer vary. Papamoschou and Brown [35] have provided an empirical expression for this for a planar shear layer:

\[
d\delta_i / dx = 0.14 \left( \frac{1 - \rho_2 \rho_1}{\rho_1 \rho_2} \right) \left( 1 + \left( \frac{\rho_2}{\rho_1} \right)^{1/2} \right) \left( 1 + \frac{\rho_1}{\rho_2} \left( \frac{\rho_2}{\rho_1} \right)^{1/2} \right)^{-1} \tag{1.1}
\]

Where \(\delta_i\) is the incompressible shear layer thickness, and \(\frac{U_2}{U_1}\), \(\frac{\rho_2}{\rho_1}\) are velocity and density ratios across the shear layer (the subscripts 1 and 2 denote fluid properties of the lower speed and higher speed flows) Equation 1.1 may then be used as a datum against which to capture the spreading rate observed at a specific Mach number (leading to given velocity/density ratios across the shear layer) with the incompressible spreading rate values for equivalent \(\frac{U_2}{U_1}\) and \(\frac{\rho_2}{\rho_1}\); any differences may then be clearly identified as compressibility effects.

Data to underpin the incompressible density ratio effects implied by Equation 1.1 have been provided by the work of Pitts [36] who used different gases in two parallel streams at incompressible speeds. The results showed a dependency of both the location of the virtual
1.3 The Influence of Nozzle Exit Conditions on Jet Shear Layer Development

origin and mixing behaviour on density ratio. Decreasing the density ratio shifted the virtual origin upstream and increased the mixing rate (spreading rate) as inferred by Equation 1.1. Increases in mixing and spreading rates were also observed in incompressible variable density axisymmetric jets by Russ and Strykowsky [37], where the jet density was reduced relative to the ambient fluid by heating. Brown and Roshko [38] used different species to achieve variable density across the shear layer and results which were again consistent with Equation 1.1; the shear layer spreading rate increased as the density of the stream of the greater velocity decreased relative to the low-speed fluid.

The first investigations to notice compressibility effects on jet development were those of Lau [39],[40] who studied perfectly expanded (i.e. no embedded compression/expansion systems within the jet) subsonic and supersonic round jets with nozzle exit Mach numbers from 0.3 to 1.7. These data showed that the length of the potential core varied with Mach numbers as:

\[
\frac{x_{pc}}{D} = aM_j^2 + c
\]  

(1.2)

This relation implies that \(x_{pc}\) increases as \(M_j\) increases, i.e. shear layer spreading rate decreases with Mach number, reaching a minimum at \(M_j = 1.4\). Further investigation showed that the minimum spreading rate was reached when the turbulent structures in the shear layer were convecting at a speed corresponding to the speed of sound in the ambient fluid. Papamoschou and Roshko [35], as noted above, were able to identify the qualitative reduction in spreading rate relative to the incompressible case using Equation 1.1 (above) and also chose to explain the Lau observation by characterising the compressibility effect in terms of a convective Mach number, \(M_c\) rather than jet Mach number (\(M_j\)). \(M_c\) was defined as the Mach number in a frame of reference moving at the same speed as the large turbulent structures in the shear layer

\[
M_c = \frac{U_2 - U_1}{a_1 + a_2}
\]  

(1.3)

Although Papamouschou and Roshko [35] studied a planar shear layer rather than free jet, the initial annular shear layer upstream of the end of the potential core is similar to this and the results of Papamouschou and Roshko [35] are valid for compressible jets. A simple curve was observed to fit the data of Papamouschou and Roshko [35] showing a decreasing growth rate of the shear layer with \(M_c\), reaching a value of 20% of the equivalent incompressible spreading rate at \(M_c = 1\). Later investigations by Goebel and Dutton [41], Clemens and Mungal [42] and Samimy and Elliot [43] into 2D compressible shear layers have all well substantiated the use of a convective Mach number as the parameter capturing the influence of compressibility on the mixing and spreading rate of compressible shear layers.
Finally, Lau [39],[40] extended his study of high speed jets to include the effect of jet temperature variations superimposed on the jet Mach number compressibility influence. As the jet total temperature was increased at constant $M_J$, the length of the potential core increased. This implied that the parameter $C$ in Equation 1.2 but not the parameter $a$ changed with temperature ratio. At $\frac{T_t}{T_0} = 1 \quad \frac{X_{pc}}{D} = 4.2$, but decreased to $\frac{T_t}{T_0} = 1.5$. A further increase of temperature ratio did not result in any further reduction in $X_{pc}$. The explanation for this effect is still unclear. Increasing jet total temperature at constant $M_J$ leads to decreased density ratio and increased velocity ratio (due to increased local speed of sound), this is in addition to changes in $M_c$ and compressibility effects, all of which influence shear layer growth in different ways. Unfortunately, literature on the effects of jet temperature on jet development is very limited; additional work is clearly required in this area.

1.3.2 The Influence of Jet Reynolds Number

Another important parameter when studying the development of the jet plume is the jet Reynolds number, based on the nozzle exit diameter and bulk velocity (area-weighted average velocity). Investigations by various workers have shown that at low Reynolds numbers ($Re_D \sim 10^4$, which may be achieved when testing at small laboratory scales) jet plume characteristics can vary significantly with jet Reynolds number. The study of near-field jet plume development of initially laminar jets by Abdel-Rahman et al. [44] issuing from smoothly contoured nozzles showed a dependency of centreline velocity decay on jet Reynolds number. Decay of the centreline velocity was expressed in the non-dimensional form:

$$\frac{U}{U_o} = B \left( \frac{D}{x-x_o} \right)$$

where $U$ is the centreline velocity at $x$, $U_o$ is the jet exit velocity, $x_o$ is the virtual origin and $B$ is the decay constant. The data indicated an asymptotic increase in $B$, from a value of 1.35 at a Reynolds number of 1430 to 6.11 at a Reynolds number of 19400, a value similar to the published values of around 6 for high Reynolds number jets ($Re_D > 100000$) [29]. The measurements of Rajaraman and Flint-Petersen [45] showed the jet mean velocity half-radius (the radius at which the velocity has decreased to one half of the centreline value) spreading rate to decrease with jet Reynolds number, decreasing from 0.2 at a Reynolds number of 2000 to a constant value of 0.16 at a Reynolds number of around 12000. This value was considerably higher than the typical value of around 0.095 for high Reynolds number flows [29], a result which was not explained by the authors.

Of particular interest in the context of mixing enhancement studies is the rate of entrainment of ambient fluid into the jet plume. The measurements of entrainment into low
1.3 The Influence of Nozzle Exit Conditions on Jet Shear Layer Development

Reynolds number \((2000 < Re_D < 80000)\) jets by Ricou and Spalding [46] also showed dependency on jet Reynolds number, with an asymptotic decrease in the rate of entrainment of ambient fluid into the jet with Reynolds number, reaching a constant value beyond a jet Reynolds number of around 25000. Pitts [47] suggested that beyond \(Re_D \approx 50000\) the influence of jet Reynolds number on jet development is minimal.

The influence of jet Reynolds number can be summarised by the comments of Bradshaw [48] who stated that when the Reynolds number of the jet is sufficiently high the flow will become independent of the jet Reynolds number and, therefore, any significant Reynolds number effects will be restricted to the near-field and be exerted through Reynolds number influences on the nozzle wall boundary layer at the separation point (nozzle lip). The work of Xu and Antonia [49] highlights such influences where jets with identical jet Reynolds numbers \((Re_D = 86000\), beyond the range of jet Reynolds number identified above as influencing jet development) but varying states of the nozzle exit boundary layers showed significant variation in the development of the near-field region with more rapid jet spreading and centreline velocity decay when the exit boundary layer was laminar compared to when it was turbulent (described in more detail in the following section). Because of the importance of the near-field in the engineering applications of relevance to this thesis, the following section investigates the influence of nozzle exit boundary layers on near-field jet development, a factor that is often overlooked in the literature pertaining to jets.

1.3.3 The Influence of Nozzle Exit Boundary Layers

Unlike global jet conditions such as Mach number, temperature ratio and Reynolds number, which are controlled entirely through rig operating conditions, the state and thickness of the nozzle exit boundary layers will vary not only with operating conditions but with different experimental facilities and nozzle geometries. Any variations in the boundary layers which form the initial regions of the jet shear layer can have significant effects on the near-field development of jet plumes.

The work of Mi et al. [50] showed the influence of jet exit boundary layers on the development of an axisymmetric, incompressible jet issuing from two nozzle types, a smoothly contoured nozzle and a long straight pipe. The nozzles were of identical exit diameter and jet Reynolds number \((Re_D = 16000)\). However, radial profiles of the mean axial velocity at exit from the pipe nozzle corresponded to a fully-developed turbulent pipe flow, whereas the jet from the smoothly contoured nozzle exhibited a top-hat profile with boundary layers following a Blasius (laminar) profile. Note that these two represent the absolute extremes of nozzle exit boundary layer thickness and state. Large variations in the spreading and centreline decay rates existed between the two nozzle exit conditions, the jet from the initially
1.3 The Influence of Nozzle Exit Conditions on Jet Shear Layer Development

A laminar nozzle exit boundary layer showed greater mixing rates, illustrated by the streamwise variation of the scalar half-radius:

- **Contraction jet:** \( r_{0.5}/D = 0.110(x/D + 1.0) \)
- **Pipe jet:** \( r_{0.5}/D = 0.102(x/D - 1.3) \)

Similar investigations of the velocity and temperature fields of jets issuing from smoothly contracting nozzles and nozzles with a parallel wall exit section at even higher Reynolds numbers \((Re_D = 86000)\) by Xu and Antonia [49],[51] also showed similar effects of exit conditions on mixing and spreading characteristics with mean-velocity half-radius spreading rates of 0.095 for the contraction nozzle and 0.086 for the pipe nozzle. Changing the state of the exit boundary layer using transition trips (Russ and Strykowski [37]) also resulted in reduction in spreading rates (by 30% compared to the laminar case) and centreline velocity decay. The variation in jet development that existed between laminar and turbulent nozzle exit boundary layers resulted from the disappearance of large-scale, coherent turbulent structures in the jets with turbulent nozzle exit boundary layers (when compared to laminar nozzle exit boundary layers), a phenomenon also observed by Hill [52]. The large scale structures form as the laminar shear layer becomes unstable, begin to roll up and are then responsible for the large-scale entrainment of fluid into the jet and more rapid mixing and greater spreading rate in the initial regions of the jet compared to those developing from turbulent nozzle exit boundary layers (as was identified by the visualisations of Brown and Roshko [38]).

The work of Russ and Strykowski [37] showed that development of the jet issuing from a smoothly contracting nozzle at a jet Reynolds number of \(10000\) with laminar exit boundary layers was also influenced by the exit boundary layer momentum thickness. Increasing the momentum thickness through the addition of parallel extensions to the nozzle in order to increase the development length of the boundary layer delayed the roll up of the shear layer by between 1.5 and 2.5 exit diameters, subsequently increasing the length of the potential core from just over four exit diameters to about six when the momentum thickness was increased from \(1/110\) to \(1/48\) of the exit diameter.

The work of Hussain and Zedan [53](62000 \(\leq Re_D \leq 112000\)) and Fondse et al. [54] \((Re_D \approx 10^5)\) showed the effects of boundary layer momentum thickness on spreading and mixing rate to be negligible. Any variation between experiments was attributed to varying initial turbulence levels [53], [55]. These results clearly contradict the findings of Russ and Strykowski[37]. It is worth noting that the jet Reynolds numbers of Russ and Strykowski's
1.3 The Influence of Nozzle Exit Conditions on Jet Shear Layer Development

experiments are within the range where the jet is influenced by the Reynolds number, therefore it is concluded that jet development in Russ and Strykowski [37] may be influenced by both the jet Reynolds number and the nozzle exit boundary layer.

Literature is also available on the influence of initial boundary layer state on mixing between two streams with various velocity ratios as encountered in coaxial jet flows. The works of Bell and Melita [56], Karasso and Mungal [57], Dimotakis [58] and Browland and Latigo [59] show that variations in the state of the boundary layer on either side of the splitter plate that divided the two streams could significantly influence the development of the free shear layer in a similar manner to the initial region of free jets, and for similar reasons.

Hill et al. [60] have provided a thorough investigation of the influence of nozzle exit boundary layer variation on jet development covering a range of nozzle scales (25.4mm, 50.8mm and 101.6mm) and a nozzle geometry consisting of a smoothly contoured contraction followed by parallel pipe sections of varying lengths. The tests were also conducted in three experimental facilities, two small-scale laboratory rigs (but with differing flow settling and turbulence management designs) and a larger-scale, higher-speed noise test facility. Nozzle exit velocities ranged from about 10m/s to 250m/s (essentially all incompressible flows). This study highlighted some of the problems associated with nozzle exhaust plume near-field behaviour in small-scale laboratory facilities due to the nozzle exit conditions described above. Hill et al. [60] found that the nozzle exit boundary layers were naturally laminar at the nozzle exits of the 25.4mm and 50.8mm nozzles, and employed transition trips at the entrance to the parallel sections to promote turbulent boundary layers. Nozzle exit boundary layers of the 101.6mm nozzle used on the larger-scale rig were found to be transitional over a jet Reynolds number range of $0.6 - 1 \times 10^6$ then naturally turbulent beyond a jet Reynolds number of $10^6$. The surveys showed that when the nozzle exit boundary layer was maintained in the turbulent state (using transition trips), the centreline velocity decay rate and potential core length remained largely unchanged with varying jet Reynolds number. Comparison between different facilities showed little sensitivity of jet plumes originating from turbulent boundary layers to variations in initial core nozzle flow turbulence levels. On the other hand, if the jet developed from nozzle exit boundary layers that were either laminar or transitional (indicated by intermittency in measured turbulence using a hot film probe), the length of the potential core was both highly unpredictable and variable, with large variations in core length (several jet diameters) existing for identical nozzle scales tested at different jet velocities and on different facilities. This was attributed to the sensitivity of the laminar boundary layer to the turbulence levels in the air supply [55] and the variation of exit boundary layer thickness with increasing jet velocity (in agreement with the findings of Russ and Strykowski [37] and Xu and Antonia [49][51]). Inspection of the facility designs
revealed that the boundary layer had no opportunity to develop upstream of the nozzle inlet in all of the three facilities used, therefore the reason for laminar nozzle exit boundary layers, even at high jet Reynolds numbers resulted from the nozzle boundary layer having insufficient length to develop and become naturally turbulent. It is a pity, however, that this study did not make any measurements at nozzle entry, nor extend the data obtained to higher jet exit velocities, for example up to the choked condition since, as will be discussed below, the high accelerations experienced by the boundary layer in the nozzle at high NPRs can exert a strong influence. The type of study conducted by Hill et al. [60] would be worth extending into the higher NPR regime.

A similar study by Lepikovsy [61] and Lepikovsy et al. [5] into the influence of nozzle exit boundary layers on heated jets for Mach numbers up to $M0.97$ and total temperatures up to 900K showed similar variations in potential core length between laminar and transitional nozzle exit boundary layers seen in the work by Hill et al.[60]. Comparison between boundary layer shape factor (shown in Figure 1.7) and potential core length for various jet temperatures and Reynolds numbers revealed that as the shape factor (an indicator of the state of the boundary layer, i.e. whether laminar or turbulent) decreased from a laminar value($\approx 2.5$) toward a turbulent one ($\approx 1.5$), the length of the potential core increased. The increase in potential core length as the boundary layer became turbulent is consistent with the results of Russ and Strykowski [37] and Xu and Antonia [49][51], who showed a decrease in mixing when the nozzle exit boundary layers became turbulent as the large-scale instabilities which were responsible for rapid entrainment of ambient fluid into jets originating from laminar nozzle exit boundary layers were not present. However, unlike the study of Hill et al. [60] (where variation in nozzle exit boundary layer state was controlled by development length of the boundary layer within the nozzle), an explanation of the reasons for the measured variation in the state of the nozzle exit boundary layer in the experiment of Lepicovksy et al. [5] (Figure 1.7) is much less apparent. A review of the Lepikovsy et al. data and experiment in Trumper [62] suggested that the experimental facility permitted sufficient development length for boundary layers to become turbulent before entering the nozzle and, therefore, any variation in the state and shape of the nozzle exit boundary layer must be a result of the development of the boundary layer within the nozzle itself. Unfortunately, like the work of Hill et al. [60], The work of Lepikovskiy et al. [5] did not extend to measurements of the boundary layer at entry to the nozzle nor its development through it, nor did the operating conditions extend into the supercritical NPRs, so there are still some unanswered questions from this work.

This review of literature on studies of nozzle exit boundary layers demonstrates clearly that jet initial conditions can influence the near-field region of the developing jet. Most significant is the influence of the state of the nozzle exit boundary layer (laminar, turbulent...
or transitional). The literature, in particular the work of Lepikovsky et al. [5] and Hill et al. [60], showed that if tests are conducted over a wide range of nozzle operating conditions where variation in nozzle exit boundary layer state occur, there may be significant variations in the development of the near-field region of the jet which may be attributable to the changing state of the exit boundary layer rather than the change in operating conditions. This is of particular significance to workers who are interested in phenomena dominated by the flow in the initial regions of the jets, such as mixing enhancement, especially when the mixing device influences jet development through interaction with the boundary layer (such as tabs [63]). Variations in jet development caused by different designs of mixing devices and those caused by changing nozzle exit boundary layer conditions would be difficult to decouple and may lead to misinterpretation of results [48]. It is important to maintain a turbulent boundary layer state at nozzle exit, as concluded by Hill. The literature, however, does not offer clear explanations as to why the nozzle exit boundary layer may become laminar or change state even when operating at high Reynolds numbers and with turbulent nozzle inlet boundary layers (most notably the work of Lepikovsky [61] and Lepikowsy et al. [5]). The following section therefore considers the factors that can influence the state of the nozzle exit boundary layer, in particular the influence of the acceleration of the boundary layer that occur within contracting nozzles at a strength which can vary widely with nozzle scale,
1.4 Factors Influencing Nozzle Exit Boundary Layers

There are two main controlling factors which determine the state of the boundary layer at nozzle exit. The first is simply whether on approaching the nozzle in the supply pipework the wall boundary layer has developed over sufficient distance in order to undergo natural transition. For boundary layers developing within straight pipes, the boundary layer can be expected to be turbulent provided that the Reynolds number of the flow, based on the diameter of the pipe, exceeds 2300. In general, laboratory test conditions would exceed this criterion. As a rough guide, if the length Reynolds number of the boundary layer (based on the distance over which it develops undisturbed) exceeds $5 \times 10^5$ [64], then transition will take place. For a nozzle with an exit velocity of $100 \text{m/s}$ this corresponds to a length of $\approx 0.07 \text{m}$, for a nozzle at the point of choking a length of only $\approx 0.02 \text{m}$ would be required for transition.

The second factor is related to processes occurring within the nozzle itself. The state of the boundary layer at nozzle exit depends both on the upstream development of the boundary layer before reaching the nozzle and on the response of the boundary layer to the flow within the nozzle itself. In full-scale engine exhaust nozzles the boundary layer is almost certain to be fully turbulent at entry to the nozzle contracting section due to the scale of the exhaust, the high velocities involved and the high levels of freestream turbulence intensity which can reach 20% within gas turbine engines [65]. In laboratory facilities designed to study jet plumes, where geometric scales are much smaller and considerable effort is invested in minimising turbulence levels through the use of turbulence management systems, this is not so certain, as was demonstrated by the studies of Xu and Antonia [49][51] and Russ and Strykowsky[37]. However, even if the nozzle inlet boundary layer is turbulent, whether the nozzle exit boundary layer is turbulent depends in addition on one further important parameter that can exert significant influence, namely on the state and development of the boundary layer, the favourable (negative) static pressure gradient that exists within the nozzle itself.

In many practical engineering flows (such as nozzles, ducts, wind tunnels and turbine cascades) boundary layers which have developed over sufficient distances to have undergone transition and become fully turbulent, are subjected to severe favourable pressure gradients. If the acceleration of the boundary layer is of sufficient magnitude and sustained over a large enough axial distance, the boundary layer may gradually revert to the laminar state, through a process known as relaminarisation or reverse transition. The process of relaminarisation
due to acceleration was first identified in the 1950s in supersonic boundary layers undergoing rapid accelerations through supersonic Prandtl-Meyer expansions and accelerating boundary layers on turbine blades. Clearly this phenomenon may influence the nozzle exit boundary layer state in laboratory small-scale model testing and should be taken into account.

Before discussing this process of relaminarisation, and the changes that occur within the boundary layer, a brief discussion of the fully turbulent, canonical zero pressure gradient turbulent boundary layer is prudent to serve as a ‘benchmark’ against which changes in the nature of accelerated boundary layers can be compared. This is presented in the following section.

1.5 The Turbulent Zero Pressure Gradient Boundary Layer

The turbulent boundary layer has been studied in extensive detail for many decades and a wealth of information exists; as such it is beyond the scope of the current investigation to perform a detailed review, only the important aspects of the turbulent boundary layer of relevance to relaminarisation are discussed below. For a detailed discussion on turbulent boundary layers the reader is directed to the work of Schlichting [66] and the detailed reviews of experimental data of Fernholz and Finley [67][68].

A fully turbulent, zero pressure gradient boundary layer is observed to display self-similarity when described in terms of local velocity and length scales which differ between two regions of the boundary layer, the inner region and the outer region.

1.5.1 The Inner Region

For the inner region, extending from the wall to approximately 10% of the overall boundary layer thickness (0.18), the flow is dominated by viscous effects and is independent of the free stream velocity and \( \delta \). The viscous scales are defined in terms of the wall shear stress, density and viscosity of the fluid such that the relevant velocity scale (the friction velocity) is defined as

\[
\text{\( u_r \equiv \sqrt{\frac{\tau_w}{\rho}} \) (1.5)}
\]

and the relevant length scale is defined as \( \frac{x}{u_r} \). The non-dimensional mean velocity becomes

\[
U^+ \equiv \frac{U}{u_r} \tag{1.6}
\]
1.5 The Turbulent Zero Pressure Gradient Boundary Layer

and the non-dimensional distance from the wall becomes

\[ y^+ \equiv \frac{u_\tau y}{\nu} \]  

(1.7)

Through dimensional analysis the non-dimensional velocity can be shown to be purely a function of the non-dimensional wall distance:

\[ U^+ \equiv f_w(y^+) \]  

(1.8)

which is referred to as the 'law of the wall'

In the very near-wall region, often referred to as the viscous (or linear) sub-layer, the contributions of turbulent stresses are negligible compared to viscous stresses as turbulent fluctuations are so small.

At the wall the wall shear stress is given by

\[ \tau_w = \mu \frac{dU}{dy} \bigg|_{y=0} \]  

(1.9)

and, since the pressure gradient is zero, a constant shear stress layer occurs near the wall such that

\[ \tau = \mu \frac{dU}{dy} = \tau_w = \text{const} \]  

(1.10)

when integrated with respect to wall normal distance, this yields the relationship

\[ U = \frac{\tau_w}{\mu} y \]  

(1.11)

or in inner wall units:

\[ U^+ = y^+ \]  

(1.12)

This linear variation of velocity with wall distance has been demonstrated to hold up to \( y^+ \approx 5 \).

At larger values of \( y^+ \) the contribution of viscous stress to the total shear stress becomes small in comparison to the turbulent stress contribution, and the dependence of the function \( f_w \) on viscosity becomes negligible. The velocity gradient in this region can be shown to be [29]:

\[ \frac{dU^+}{dy^+} = \frac{1}{\kappa y^+} \]  

(1.13)
1.5 The Turbulent Zero Pressure Gradient Boundary Layer

Figure 1.8: Turbulent Boundary Layer Profile Using Inner-Wall Scaling

which integrates to form the familiar ‘Logarithmic Law of the Wall’

\[ U^+ = \frac{1}{\kappa} \log y^+ + B \]

which is widely accepted as a universal curve for the turbulent part of the inner region of the boundary layer at high Reynolds numbers. Pope [29] reports that the experimental values available in the literature for the constants \( \kappa \) and \( B \) are in the region of 0.41 and 5.2 respectively, but vary between different works by around 5%.

Between the viscous-dominated and logarithmic region there exists a ‘buffer region’ where there is transition between the domination of viscous and turbulent stresses. The existence of both linear and logarithmic regions has been captured in many experiments as illustrated in Figure 1.8.

1.5.2 The Outer Region

Beyond \( y^+ \approx 50 \) the assumption that the influence of viscosity becomes negligible suggests that at even larger distances from the wall the mean velocity, \( U \), becomes a function of the global boundary layer size and velocity scales only i.e. \( U_\delta \) (the velocity at the edge of the boundary layer, typically defined as 0.99\( U_\infty \)) and \( \delta \), and is independent of viscous scales.
1.6 Pressure Gradient Effects on Boundary Layers

Von Kármán \[69\] showed that this implies

\[
\frac{U_\delta - U}{u_r} = f_d \left( \frac{y}{\delta} \right)
\]  \(1.15\)

where \(U_\delta - U\) is referred to as the 'Velocity Defect'. Equation 1.15 shows that the velocity defect, non-dimensionalised with respect to the friction velocity is a function of \(y/\delta\) only.

At high Reynolds numbers there is an overlap between the inner and outer regions, therefore the velocity defect law of Equation 1.15 must be logarithmic in order to match the logarithmic nature of the inner layer:

\[
\frac{U_\delta - U}{u_r} = \frac{1}{\kappa} \log \left( \frac{y}{\delta} \right) + C
\]  \(1.16\)

Coles \[70\] noted that in the outer layer the velocity profile departs from the logarithmic profile, especially in the presence of a pressure gradient, with the deviation resembling the shape of a wake. He proposed the law of the wake, which can be added to the inner law of the wall such that

\[
U^+ = \frac{1}{\kappa} \log y^+ + B + \frac{2\Pi}{\kappa} f \left( \frac{y}{\delta} \right)
\]  \(1.17\)

where \(\Pi\) is Coles wake function which is dependent on the flow, in particular the pressure gradient.

Now that the canonical fully turbulent boundary layer has been described, it is possible to turn to a discussion on the influences that the favourable static pressure gradient that exists within propulsion nozzles can have on such a boundary layer.

1.6 The Effect of a Favourable Pressure Gradient on Turbulent Boundary Layers

In his review article on the subject of relaminarisation Sreenivasan \[71\] stated that:

'relaminarisation is a gradual process accompanied by drastic changes in the structure of the boundary layer.'

The drastic changes referred to above are the thickening of the viscous sub-layer (described by Narashima and Sreenivasan \[72\] as the formation of a sub-laminar boundary layer), departure of inner region velocities from the familiar logarithmic region of the universal law
1.6 Pressure Gradient Effects on Boundary Layers

of the wall (as shown in Figure 1.9 below), a decay of Reynolds shear stress to levels much lower than in the zero pressure gradient boundary layer, as well as changes in boundary layer parameters such as the shape factor increasing toward a laminar value, increased local skin friction coefficient (as shown in Figure 1.10 below) and a reduction in heat transfer coefficient [73], to name but a few.

Figure 1.9: Variation of the Mean Velocity Profile During Relaminarisation (Case 2, Warnack [74])

Sreenivasan [71] divided the process of relaminarisation caused by acceleration of the flow into several regions, as shown in Figure 1.11. In the first region pressure gradient effects on the boundary layer are small. In this region the near-wall mean velocity profiles still follow that of the universal law of the wall. If the magnitude of the pressure gradient increases beyond that experienced in the initial region, the acceleration begins to have significant effects on the structure of the boundary layer. The thickness of the outer wake region (described in the previous section) decreases and the velocities in the wall region no longer follow the standard logarithmic law of the wall, overshooting the standard log-law line starting at the upper edge of the buffer region \((y^+ \approx 30\), Figure 1.9). Sreenivasan, [71] further divided this region into an equilibrium region where turbulence production and dissipation were still approximately equal, and a non-equilibrium region where they were not. These regions are often referred to as 'laminarescent', as the boundary layer is still turbulent but shows characteristics of non fully-turbulent flows. Sreenivasan [71] was keen to point out
that transition from fully turbulent to laminarescent does not guarantee that the flow will eventually completely revert to the fully laminar state, instead it is a stage on the route to full relaminarisation. The final region is the process of relaminarisation itself. If the pressure gradient is large enough and sustained, the velocity profile reverts fully to a laminar state and the turbulence within the boundary layer prior to relaminarisation becomes inactive [75], and does not influence the velocity profile.

The process of relaminarisation has been studied in considerable detail over the last four decades, both experimentally and numerically, with particular focus on the causes of relaminarisation, with attempts to provide simple parameters that identify whether a boundary layer is susceptible to relaminarising. Attention is now directed to a brief review of these previous studies, starting with the mechanism of relaminarisation.

1.6.1 Mechanism of Relaminarisation

Launder [76] summarises relaminarisation as the inability of the boundary layer to adjust rapidly to the addition of a large favourable static pressure gradient. The acceleration of the outer region of the boundary layer brings about a large increase in momentum in this region which, in turn, results in greater viscous shear stresses. If the pressure gradient is maintained for sufficient distance, the viscous stresses become proportionally larger than the turbulent shear stresses due to the inability of the turbulent structures to respond to the acceleration as quickly as the mean flow [77]. As it is the local gradient $\frac{\partial u}{\partial y}$ that is responsible for turbulence production, not the pressure gradient, the turbulence in the outer region (where the spatial velocity differences are small) is unable to respond rapidly to the change in pressure; instead there is a global acceleration of the fluid with the gradient $\frac{\partial u}{\partial y}$ remaining unchanged. This response of turbulence to a favourable pressure gradient has been demonstrated by the Large Eddy Simulation of Piomelli [78]. The simulation showed that the turbulence within the boundary layer does adjust to the pressure gradient (with increases in the absolute level of turbulent kinetic energy) but the rate of increase is lower than the increase of the mean kinetic energy in the main flow such that the turbulence is said to 'lag the main flow'. The structure parameter, $a_1 = \frac{\langle w'_y w'_z \rangle}{\langle w'_z \rangle^2}$ (described by Piomelli [78] as a measure of the efficiency by which the turbulence can extract Reynolds shear stress from the turbulence kinetic energy) showed a decrease in the near-wall region, indicating a decreased contribution to the turbulent stresses. Eventually the rate of dissipation exceeds the rate of production and the turbulence will begin to decay. Bradshaw [79] used an eddy Reynolds number to explain the mechanism for relaminarisation. The eddy Reynolds number
1.6 Pressure Gradient Effects on Boundary Layers

Figure 1.10: Variation of Boundary Layer Parameters During Relaminarization (Case 2, Warnack [74])

Figure 1.11: The stages of relaminarisation [71]
1.6 Pressure Gradient Effects on Boundary Layers

is defined as

\[ Re_l = \frac{\ell (\frac{z}{\nu})^{\frac{1}{2}}}{\nu} \]  

(1.18)

where \( \ell \) is the dissipation length parameter defined as

\[ \ell = \frac{(\frac{z}{\nu})^{\frac{3}{2}}}{\epsilon} \]  

(1.19)

At a critical value of the eddy Reynolds number, the length scales of the energy-containing structures and the range of eddy sizes where dissipation takes place overlap, and this leads to rapid dissipation of the energy-containing turbulent structures.

Narashima and Sreenivasan [72] concluded that reversion to the laminar state in highly accelerated boundary layers is caused by

"the domination of pressure forces over the slow responding Reynolds stresses in an originally turbulent flow, accompanied by the generation of a new sub-laminar boundary layer stabilised by the favourable pressure gradient"

1.6.2 Behaviour of the ‘Relaminarised’ Boundary Layer

The question of whether a turbulent boundary layer can fully revert to the laminar state is answered in the classical work of Launder [75] on the relaminarisation of a two-dimensional incompressible boundary layer. Launder [75] cites that for a laminarised boundary layer

"A state can be reached where this boundary layer does not differ in any other gross features from that of a laminar boundary layer"

This statement is corroborated by experimental evidence. Boundary layer surveys show that the mean velocity profile closely follows that of the Blasius laminar boundary layer on a flat plate and measured turbulent shear stresses at the end of the acceleration region are negligible compared to viscous stresses. Narashima and Sreenivasan [80] defined the process of relaminarisation to be complete once the effect of the Reynolds shear stresses on the development of the boundary layer became negligible. Of particular significance when considering the influence of laminar nozzle exit boundary layers on jet plume development is the comparison of the behaviour of a relaminarised boundary layer to a naturally laminar boundary layer. Launder subjected a relaminarised boundary layer and a naturally laminar boundary layer to an adverse pressure gradient (10° diffuser). Both boundary layers were
seen to separate at the same location, and showed similar behavior on re-attachment and transition to the turbulent state. Therefore, it can be expected that the shear layer of a free jet originating from a relaminarised boundary layer will behave in a similar manner to one originating from a naturally laminar one.

Although a clear end to the process of relaminarisation can be defined, the same is not true of the beginning, or 'onset' of relaminarisation. This is because, although relaminarisation causes catastrophic changes in the nature of the boundary layer, the process is gradual and, therefore, hard to define precisely when it has begun. This has given much scope to workers to attempt to define the optimum criterion for the onset of relaminarisation. A review of such studies is presented below.

1.6.3 Previous Experimental Work on Relaminarising Boundary Layers

Patel and Head [81] presented measurements of a highly accelerated axisymmetric boundary layer at low speed. Acceleration of the boundary layer was achieved by using an axisymmetric centrebody mounted within the pipe. The location of the centrebody was varied so as to increase the distance over which the initial boundary layer could develop in a negligible pressure gradient. The authors used a modified form of the logarithmic law of the wall which took into account the non-constant shear stress distribution in the near-wall region of flows with non-zero pressure gradients. Departure of the mean velocity profile from this modified law of the wall (overshooting) was taken to denote the beginning of relaminarisation. This point was then identified with a particular value of the non-dimensional pressure gradient parameter $\Delta_p$ defined as:

$$\Delta_p = \frac{\nu}{\rho u^2} \frac{dp}{dx}$$

The critical value of $\Delta_p$ in the experiments was identified as less than $-0.0245$, suggesting that a value of $\Delta_p$ below this value might be used as a criterion to identify the onset of relaminarisation. The overshoot occurred at the same $\Delta_p$ for two initial boundary layer momentum thickness Reynolds numbers. Increases in the shape factor also occurred at the same value of the pressure gradient parameter ($\Delta_p \approx -0.0235$), but occurred slightly earlier than the departure from the logarithmic law of the wall. Similar breakdown in the law of the wall was observed by Badri Narayanan and Ramjee [82] once $\Delta_p$ had reached $-0.02$.

Fiedler and Head [83] presented an investigation of intermittency in two-dimensional turbulent boundary layers using hot-wire and optical techniques. Intermittency describes the switching of the flow between turbulent and non-turbulent states and is defined as the
ratio of the time for which the flow is turbulent \( (\gamma) \), \( \gamma = 1 \) thus represents a fully turbulent flow. Their results showed that if a boundary layer is accelerated the lines of constant intermittency move toward the wall indicating that greater proportions of the boundary layer are non-turbulent. If the acceleration is of sufficient magnitude, the boundary layer became intermittent across its whole thickness once the boundary layer has reverted to the laminar state, thus suggesting intermittency as a possible criterion to identify reversion. Intermittency measurements by Blackwelder and Kovasznay [84], Escudier et al [85] and Ichimiya et al. [86] showed similar results.

The most widely used parametric criterion for the onset of relaminarisation is the acceleration parameter ‘K’ of Launder [75] defined as:

\[
K = \frac{\nu}{U_\infty^2} \frac{dU_\infty}{dx} = -\frac{\nu}{\rho U_\infty^3} \frac{dp}{dx}
\]

and modified by by Nash-Webber and Oates [87] and Goldfeld [88] to use viscosity at the wall for compressible flows to

\[
K_c = \frac{\nu_w}{U_\infty^2} \frac{dU_\infty}{dx} = -\frac{\nu_w}{\rho_w U_\infty^3} \frac{dp}{dx}
\]

It should be noted that K may be regarded as the inverse of a Reynolds number,

\[
K = Re^{-1} = \left( \frac{UL}{\nu} \right)^{-1}
\]

with the length scale derived from the pressure gradient (which is determined by the flow geometry)

\[
L = \left( \frac{1}{U_\infty} \frac{dU_\infty}{dx} \right)^{-1} = \left( \frac{1}{\rho U_\infty^2} \frac{dP}{dx} \right)^{-1}
\]

and is related to \( \Delta_p \) such that

\[
K = -\Delta_p \left( \frac{\epsilon_f}{2} \right)^{-\frac{3}{2}}
\]

The acceleration parameter attempts to quantify the beginning of relaminarisation based on the conditions required for the velocity profile to over-shoot the logarithmic region of the universal law of the wall. Its popularity in the reviewed literature stems from its ease of definition, being solely comprised of easily measured freestream quantities, compared to other parameters (such as \( \Delta_p \)) which require skin friction and shear stresses which are often difficult to measure. Launder’s analysis [75] suggested a critical value of order \( 3 \times 10^{-6} \). The
review of the available literature in Trumper [62] showed reported critical values ranging from $2.7 \times 10^{-6}$ to $3.5 \times 10^{-6}$.

Badri Narayanan and Ramjee [82] presented a series of tests on planar two-dimensional boundary layers subjected to favourable pressure gradients. Seven conditions were presented with varying axial velocity variation and initial boundary layer Reynolds numbers. Hotwire measurements of axial velocity fluctuations showed that major decreases in magnitude occurred only when the boundary layer momentum thickness Reynolds number decreased below 400 suggesting a reduction of $Re_\theta$ to $300 \pm 100$ as a parametric criterion for relaminarisation. However, Sreenivasan [71] and the experimental results of Warnack and Fernholz, [74] Warnack [89], Blackwelder and Kovasznay [84] and the Large Eddy Simulation (LES) of Piomelli et al. [78] do not support the critical Reynolds number criterion since values of the boundary layer momentum thickness Reynolds number at onset of relaminarisation were greater than 400 (fourfold in Blackwelder’s experiment). The review of incompressible boundary layer data by Fernholz and Finley [67] showed that below $Re_\theta \approx 350$ a logarithmic region in the boundary layer cannot be identified and may be regarded as the minimum Reynolds number for a self-sustaining fully-developed turbulent boundary layer. The results of Badri Narayanan and Ramjee [82] may therefore result from their very low initial Reynolds numbers. Sreenivasan [71] suggested that the occurrence of relaminarisation at $Re_\theta$ less than 400 is purely a product of the limited Reynolds numbers achievable in ‘academic’ wind tunnels, and questioned the applicability of such results to high Reynolds number flows. Indeed, the use of boundary layer trips in some of the literature (Ichimiya [90], Ichimiya et al [86], Fernholz and Warnack [91], Warnack and Fernholz [74]) suggests that there was often insufficient distance for a turbulent boundary layer to naturally develop fully. Sreenivasan [71] argued that it is essential to have an initial fully developed boundary layer in order to understand the effects of pressure gradient on turbulent boundary layers. A review of boundary layer conditions prior to acceleration [62] showed the majority of initial boundary layer Reynolds numbers ($Re_\theta$) were around 500 to 1500.

These comments highlight the limitations of the experimental data published so far and reviewed above. The narrow range of initial conditions ($Re_\theta$) does not represent the range that may be encountered in the high Reynolds number flows associated with high speed jets. In such flows momentum thickness Reynolds numbers may be orders of magnitude greater and vary with changing operating conditions due to large changes in velocity, density and viscosity in the region where the boundary layer is developing. The data do not currently permit the assessment of the influence of initial conditions, and parameters such as the acceleration parameter do not take any account of the history of the boundary layer prior to acceleration. The parametric criteria of Launder, Patel and Head, and Narashima and Sreenivasan suggest a link between Reynolds number and susceptibility to relaminarisation,
through skin friction. The widely used acceleration parameter, \( K \) (criticised by Sreenivasan [71] as it does not consider the state of the boundary layer on which the pressure gradient acts, being comprised solely of free stream conditions) assumes a value of skin friction coefficient \( c_f = 0.004 \) for the sake of simplicity (as explained by Launder [75]).

Unfortunately, little information on relaminarisation at high Reynolds numbers is available. Narashima and Sreenivasan [72] noted that the experimental investigation of boundary layers within supersonic convergent-divergent nozzles by Nash-Webber and Oates [87] identified a possible link between the compressible acceleration parameter \( K_c \) (Equation 1.22) and the initial boundary layer momentum thickness Reynolds number. Relaminarisation, identified as the point of minimum shape factor, corresponded to a value of \( K \) of about \( 5.7 \times 10^{-6} \) at decreasing to \( 2.3 \times 10^{-6} \) as the initial \( Re_{th} \) approximately doubled (from \( Re_{th} \approx 2000 \) to \( Re_{th} \approx 5000 \)). The authors aimed to provide a design guide for laminarising nozzles, and using their own data and those of other investigators produced a criterion for relaminarisation that was independent of skin friction as this is difficult to measure experimentally. They suggested, albeit tentatively, that relaminarisation could be predicted from the variation of momentum thickness Reynolds number with acceleration parameter through the nozzle which encompassed the history of the development of the boundary layer, referred to as the ‘shear layer trajectory’ by the authors. The region where relaminarisation could be expected was bounded by the curve given by

\[
K = 1.2 \times 10^{-6} + 1.1 \times 10^{-10} Re_{th} + 10^{-13} Re_{th}^2
\]  

(1.26)

Narashima and Sreenivasan [72], however, pointed out that this conclusion appears to contradict their own findings as the critical value of the acceleration parameter would increase with increasing \( Re_{th} \), but their findings implied that relaminarisation occurs within an enclosed \( K-Re_{th} \) space, not necessarily at the boundary and that a \( K-Re_{th} \) relationship at the point of onset could not be inferred from their results. In order for relaminarisation to occur Nash-Webber and Oates argued that ‘sustained entry’ into the region above the curve was required for relaminarisation, but failed to detail how long the acceleration would need to be sustained for relaminarisation.

A similar investigation of supersonic nozzle relaminarisation by Back et al. [92],[93] also show a dependency on initial \( Re_{th} \) for high speed, accelerated boundary layers within a convergent-divergent nozzle with initial momentum thickness Reynolds numbers up to 13000. Observations of the onset of relaminarisation agree with the parameter:

\[
K \left( \frac{c_f}{2} \right)^{-\frac{1}{2}} = 0.040
\]  

(1.27)
which agrees with Launder's [75] analysis of the departure from the logarithmic law of the wall. It can be concluded from the experimental evidence above that relaminarisation may be influenced by initial $Re_\theta$. Variations in initial $Re_\theta$ in small-scale, low-speed experimental facilities may not be of sufficient magnitude to observe changes in the critical value of $K$ and, therefore, susceptibility of boundary layers to relaminarisation. However, in larger-scale and high speed facilities where variation in initial $Re_\theta$ may be large, the susceptibility of the nozzle boundary layer to relaminarise may vary significantly with operating conditions. It is clear that further work on the effects of initial $Re_\theta$ is required in order to confirm the observations of Nash-Webber and Oates [87] and Back et al.[92],[93].

1.6.4 Previous Numerical Work on Relaminarising Boundary Layers

The phenomenon of relaminarisation has also been investigated numerically using RANS CFD, with models developed to predict low Reynolds number flows. The motivation behind the development of the Low Reynolds Number (LRN) form of the $k-\epsilon$ turbulence model of Jones and Launder [94] was indeed the ability to predict relaminarisation. Simulations of a turbulent sink flow where the acceleration parameter remained constant ($K = 2.2 \times 10^{-6}$[13]) predicted the overshoot of the mean velocity above the universal log-law line although not to the same extent as measured experimentally. Predictions of skin friction coefficient were within 10% of experimental values, within the bounds of experimental uncertainly at the time. Predictions of the heat transfer experiments of Moretti and Kays [73] mirrored closely the variation of the Stanton number through the accelerated region, most importantly the rapid decrease in heat transfer associated with relaminarisation but predicted a more rapid increase in heat transfer than observed experimentally once the acceleration was relaxed.

The performance of different turbulent models when simulating the relaminarising flows of Patel and Head [81] and Badri-Narayanan and Ramjee [82] was discussed as part of a review of turbulence models for near-wall and LRN flows by Patel et al.[95]. The models tested included the LRN $k-\epsilon$ models of Launder-Sharma (LS)[96] (derived from the Jones and Launder model[94]), Chien (CH)[97] and Lam-Bemhorst (LB)[98] which had been shown to predict LRN flat plate boundary layers with reasonable accuracy. Comparisons of skin friction coefficients showed that all of the models predicted the results of Patel and Head, with the LS model slightly under-predicting experimental values. For the work of Badri-Narayanan and Ramjee, where the acceleration was much more severe ($K = 7 \times 10^{-6}$) only the LS model was capable of satisfactorily predicting the skin friction variation, the other models only capturing the variation qualitatively. Despite the limited data in terms of turbulence parameters and velocity profile shapes, the authors concluded that the LS
model, and to a lesser extent the models of CH and LB produced comparable results with experimental data but required further improvement if they were to be used to predict flows at low Reynolds numbers. Viala and Aupoix [99] tested the ability of a wide range of turbulence models to predict relaminarisation. They found that LRN models that used damping functions that were independent of wall distance (such as the LS model) were capable of predicting relaminarisation whereas ones that involved the use of wall distance (such as $y^+$) did not without additional corrections. Simulations of the experiments of Blackwelder and Kovasznay [84] showed that the LS model predicted the increase in shape factor well, and performed well when predicting the behaviour of the turbulence up to the end of the acceleration. Beyond the accelerated region the LS model over-predicted the shape factor and under-predicted the skin friction coefficient. Predictions of the work of Moretti and Kays showed the LS model most closely following the variations of heat transfer. Most significant to the current work was the simulation of the nozzle flows of the Nash-Webber and Oates experiments [87]. Unfortunately, little experimental detail was reported, so initial conditions were based on reported initial boundary layer momentum thickness and shape factor. The results showed the LS models and a corrected form of the CH model captured the rise in the kinematic shape factor as the boundary layer relaminarised. The authors also reported that the corrected CH model also predicted the skin friction well, whereas the LS model underpredicted the skin friction. Unfortunately, the degree of under prediction was not reported and no details of mean velocity or turbulence statistics were reported.

The current literature on the modelling of relaminarisation in compressible flows is very limited, attention has mainly been focused on incompressible scenarios where experimental data are more plentiful. At present, only one study of relaminarisation of compressible boundary layers which are expected within high speed nozzles is available within the open literature (Viala and Aupoix [99]). As a result little understanding has been gathered on the ability of the low Reynolds number turbulence models to capture relaminarisation under these conditions.

1.7 Summary

The influence of nozzle conditions and nozzle exit boundary layers has been widely reported in the current literature. All work has highlighted the important role that nozzle exit boundary layers can play in the development of the near-field region of the jet plume, a feature that cannot easily be controlled through flow conditions such as NPR.

The review of the current literature has also highlighted several deficiencies. Most obvious is the lack of information on the influence of nozzle exit boundary layers on the development
of jets for operating conditions which would be encountered in practical aerospace exhaust
nozzle applications, particularly close to or above the critical NPR. Only two studies have
been identified which detailed both the nozzle exit boundary layers and the development
of the near-field region of the jet for realistic operating conditions: the work by Hill et al.
[60] and the work by Lepikovsky et al. [5]. Although the former covered a wide range of
Reynolds numbers, nozzle scales and operating conditions, all tests were limited to subsonic
(and essentially incompressible) velocities. The measurements of nozzle exit boundary layers
by the latter included jet Mach numbers up to 0.97 but did not enter the underexpanded
regime. However, the data of Lepikovsky et al. [5] are somewhat puzzling as the state of the
nozzle exit boundary layer was often laminar when it could be expected that the boundary
layer at inlet to the nozzle was turbulent [62]. Therefore it is not known whether the nozzle
exit boundary layer was naturally laminar, or if it had reverted from the turbulent state
due to processes within the nozzle. It is also not known whether such changes in nozzle exit
boundary state would effect the development of the near-field region of the jet compared to
one originating from a nozzle with a naturally laminar exit boundary layer. The work of
Lepikovsky et al. [5] highlights another deficiency in the studies of the influence of nozzle
exit boundary layers on jet development. None of the studies identified in the literature
review detailed the development of the boundary layer within the nozzle itself, which would
shed light on any trends in the behaviour of nozzle exit boundary layers with varying NPR
and how they may influence jet development.

The literature reviewed has also demonstrated that the large favourable static pressure
gradients which occur within propulsion nozzles will result in a possible relaminarisation
of a turbulent boundary layer if these are of large enough magnitude and sustained for
sufficient distance. A few studies that examined the development of compressible boundary
layers through the nozzle are available (Back et al. [92][93] and Nash-Webber and Oates [87])
but their operating conditions and experimental data were limited and the nozzles tested
(convergent-divergent) were more representative of supersonic wind tunnels than practical
engine exhaust nozzles which are the focus of the current study. Clearly a knowledge of
the boundary layer upstream of the nozzle inlet and its development through the nozzle
would be a worthwhile addition to any study on nozzle exit boundary layers and near-field
jet development. Currently there are no such studies within the published literature. One
reason for such omissions may be the difficulty of performing measurements of the boundary
layer within the nozzle due to internal nozzle measurements being impractical (due to hostile
conditions or access problems such as the requirement of optical windows for laser-based
measurement techniques, or access and sealing for mechanical probes). Boundary layers may
also be of insufficient thickness to resolve using conventional measurement techniques due to
insufficient development length of the boundary layer or the influence of a favourable pressure
1.8 Aims and Objectives

The aims and objectives of the present investigation stem from the summary of the literature provided here. Although it was identified in the literature review that the effects of total temperature on jet near-field development requires further study, the wide range of aims and objectives dictate that the present work be limited to unheated nozzle and jet flows. As such, the effects of total temperature, particularly on the nozzle exit boundary layer, are beyond the scope of this work.

The objectives of the current work were to assess the influence of nozzle scale and operating conditions on nozzle exit boundary layers and to detail the initial development of the jet plume near-field through a combination of experimental and numerical techniques for an extended range of NPRs compared to existing studies. The main aims can be divided thus,
1.9 Thesis Structure

The rest of this thesis is structured as follows:

Chapter 2 describes the approach used in the experimental work to be reported and includes details of the experimental facilities, measurement techniques, data reduction and sources of error. Attention is also placed on the location of measurements and the simplification of data reduction and comparison through the use of kinematic parameters to remove the influence of compressibility on the development of the boundary layer through the nozzle.

Chapter 3 presents the numerical approach adopted in this thesis and includes the governing equations, modelling of the turbulence using the Launder-Sharma low Reynolds number turbulence model and a description of the flow solver used. Finally, the Launder-Sharma model is validated against a low Reynolds number, fully developed turbulent pipe flow and a relaminarising axisymmetric boundary layer.

Chapter 4 presents the experimental results of nozzle inlet and exit boundary layer measurements for a wide range of operating conditions and for two different nozzle scales and geometries (conical nozzles with and without parallel wall extension pieces). Jet plume near-field measurements are also presented to assess the influence of nozzle exit boundary layers on the development of the free jet.

Chapter 5 presents the results of the numerical prediction of boundary layer development within the nozzle. The influence of the addition of parallel-walled extension pieces on the boundary layer and nozzle flow field is also studied and presented.

The final chapter presents the primary conclusions and suggests further work in the area of nozzle boundary layer development and its influence on the near field region of high speed jet plumes.
Chapter 2

Experimental Facilities and Methodology

As identified in the review of current literature in Chapter 1, the range of experimental conditions has been limited mainly to low speeds and low boundary layer momentum thickness Reynolds numbers. Although published data do exist for high speed flows the data are sparse and do not permit definite conclusions on boundary layer relaminarisation and the influence of the boundary layer on jet near-field development to be made with any confidence.

For this reason, an experimental investigation was undertaken in the High Pressure Nozzle Test Facility at Loughborough University (to be discussed below) in order to catalogue the state and shape of nozzle exit boundary layers with varying nozzle operating conditions. This required new additions to the facility to permit the measurement of boundary layer profiles both within the delivery duct prior to entry to the nozzle and at the nozzle exit itself. The former would also be useful in providing inlet conditions for the numerical predictions of boundary layer development through the nozzle (to be described in Chapter 5) and the latter would allow assessment of the effects of the nozzle acceleration on the boundary layer. The methods adopted (primarily using pneumatic probes) to perform such measurements are described in Section 2.2.

As Mach numbers increase, the use of intrusive measurement techniques, such as pneumatic probes, becomes more difficult and less accurate due to the formation of normal shock waves in front of the probe caused by local flow acceleration due to the displacement effect of the probe [100]. The use of the non-intrusive Laser Doppler Anemometry (LDA) technique avoids local shock wave formation. Unlike probe-based techniques, it also permits the measurement of individual components of velocity and turbulence characteristics such
as RMS velocities and Reynolds shear stresses, which are important indicators when studying relaminarisation of turbulent boundary layers, as discussed in Section 1.6.1; hence this measurement technique was also adopted in the present experimental work.

This chapter documents the approach to the experimental component of the present thesis, starting with a description of the experimental facilities used in the current investigation (Section 2.1). The basic techniques and required facility modifications are described in Sections 2.2 and 2.3. Data reduction methods are detailed in Section 2.4, with particular attention being paid to the effects of compressibility to allow comparison over a wide range of nozzle operating conditions (NPR). Finally, accuracy and sources of error for both probe and LDA based measurements are discussed in Section 2.5.

2.1 The Test Facility

All of the experimental investigations were conducted using Loughborough University's High Pressure Nozzle Test Facility (HPNTF). The facility was introduced in 1994 to permit the study of single or dual stream (primary/secondary) supersonic jet flows for a wide range of nozzle scales, geometries and operating conditions, up to pressure ratios of 5. The range of operating conditions was further extended to include high temperature flows by the addition of an inline combustor in 1998 to study temperature/density effects on jet plume development, allowing jet total temperatures of up to 900K in the primary stream. The following section contains a description of the layout and operation of the facility.

2.1.1 Air Supply

A schematic of the air supply system is given in Figure 2.1 and the individual components of this are briefly described here.

The air supply is provided by a Bellis and Morcom, two-stage, W type, reciprocating compressor, powered by a 380HP electric motor, located in a separate building. The compressor delivers 0.8kg of air per second at a maximum gauge pressure of 13.8 Bar. After leaving the compressor, the air is passed through an intercooler to reduce its temperature, a filter to remove any particulate matter and an oil separator which removes any oil and condensate from the air. In order to reduce the moisture content of the air (which could otherwise condense in the jet, potentially damaging measurement equipment and interfering with some measurement techniques such as LDA) the air is then dried. The compressed air is passed through a dual chamber, Hankinson model DK1900ce, desiccant type, compressed air dryer which reduces the dew point to $-40^\circ$C. The air is then stored in a system of 8
interlinked air receivers with a total capacity of 110$m^3$ located outside of the main test cell building. The air receivers act as a pulsation damper, removing any pressure fluctuations attributable to the reciprocating compressor and dryer operation, as well as acting as a reservoir of air when the facility is operating in 'blow-down' mode. This latter mode of operation is necessary when the nozzle size and NPR condition leads to an air mass flow rate greater than the compressor maximum steady flow supply. The total temperature of the air supply was observed to be constant and equal to the ambient temperature; this is due to the large surface area of the receivers and the residence time of the air in the receivers allowed thermal equilibrium with their surroundings to be established [101]. A working pressure of 10 Bar can be achieved within 15 minutes. One of the air receivers can be isolated from the others once fully charged by means of a servo-controlled, three-way valve; this receiver can then be used to provide a tertiary air supply for novel mixing devices such as fluid tabs, although this feature was not used in the current tests.

2.1.2 Air Supply Pressure and Mass Flow Rate Control

The compressed air is ducted to the test cell containing the nozzle rig via a 6 inch pipe, connected to all of the air receivers by a manifold. The air passes through two valves located above the test cell (Figure 2.1). The first is a hand-operated globe valve which provides a means of isolating the air supply for maintenance purposes. The second valve is a pneumatically-controlled, 4 inch G4 valve. This valve provides baseline pressure (and, hence, mass-flowrate) control, reducing the supply pressure from the level in the air receivers (as high as 15Bar (absolute)) to a level closer to desired nozzle test condition (typically 5 Bar (absolute)). The valve is controlled from within the test facility control-room using an independent (shop air) air supply.

Once the air has reached the test cell (Figures 2.2 and 2.3) it passes through a second hand-operated globe valve, used to isolate the nozzle test rig in the test cell. The flow is then split into two streams, one to supply a primary nozzle and the other (if required) to supply a secondary nozzle (for example the outer flow stream if a coaxial exhaust nozzle configuration were under test). The main mass flow and nozzle pressure control system comprises a pair of computer-controlled, pneumatic valves. For this investigation, only the primary nozzle stream was used and is described below.

The primary stream control comprises a Spirax-Sarco type C, 4 inch valve combined with a Spirax-Sarco Smart Positioner SP12 and a programmable PID controller which is located on the main control panel in the test cell control room (illustrated in Figure 2.4). The main function of this valve is to set and maintain the total pressure of the air supplied to the primary nozzle. Control of the nozzle supply total pressure can be achieved in two ways. The
supply pressure can be controlled automatically by setting the desired operating pressure and using the PID controller to maintain this. Feedback to the PID controller is provided by a Pitot probe mounted in the middle of the delivery pipe, approximately 0.5m downstream of the primary control valve shown in Figure 2.3, combined with a calibrated Huba Control type 691, 6 Bar pressure transducer. The supply pressure can also be controlled manually by opening the valve a fixed percentage using the controller in manual mode. In this mode the operator relies on the pneumatic G4 valve to maintain the constant supply pressure. The valve is also used as the main isolating valve during testing to stop the airflow between tests.

2.1.3 Combustion System

After the main control valve the primary air then enters a combustor which can be used to raise the total temperature of the flow for high temperature tests. All of the primary air flow passes through the combustor for both heated and unheated test conditions.

The combustion system comprises a single can-type combustor from the Rolls-Royce Spey and Tay family of gas turbine engines, located in a purpose-built pressure vessel. The combustor is fueled using JET-A1 aviation kerosene supplied from a location outside of the test cell building. Flow temperature downstream of the combustor is controlled manually by altering the overall fuel/air ratio entering the combustor, by varying the speed of the fuel supply pump from within the control room. Total temperatures are measured at the combustor exit and the nozzle delivery duct (shown on Figure 2.3) by calibrated K type junction thermocouples. Additional thermocouples are mounted at the combustor exit to monitor skin temperatures and at several locations on the rig as part of a safety system to shut down the combustor in the event of temperature limits being exceeded (900K maximum). The combustor was not in use in any of the tests reported in this thesis.

2.1.4 Air Delivery to the Test Nozzle

After leaving the combustor, the flow enters a transition piece to return the duct cross-section from a 36° annular arc at the combustor exit back to a circular cross-section and into a 0.155m internal diameter duct of 1.166m in length which is insulated. The flow then passes through a 4 : 1 contraction into the final, uninsulated delivery duct which has a total length of 1.37m and is 0.078m internal diameter. The internal diameter of the delivery duct is reduced to 0.075m just upstream of the nozzle attachment point. A carefully machined groove on the outside of the nozzle holder extension duct allows for the attachment of various test nozzles using a series of grub screws distributed equally around the circumference.
2.1 The Test Facility

After passing through the test nozzle, the flow enters the jet plume measurement region, which extends for 2m downstream of the nozzle exit. Measurements are conducted in a nominally static ambient environment, with the fluid required for entrainment into the jet entering the test cell through a flow conditioner, designed to reduce noise breakout from the test cell, which is located on the roof of the facility. Finally, the jet plume enters a detuner which attenuates jet noise and exhausts the jet fluid to atmosphere.

2.1.5 Test Nozzle Design

Several test nozzles have been used in the present investigation. The main geometry considered was a simple convergent, conical design based broadly on the internal geometry of the BAE SYSTEMS Hawk jet trainer, with an internal contraction half angle of 11° (Figure 2.5). All nozzles have the same inlet diameter of 0.075m, but differing nozzle exit diameters of 0.048m and 0.06m and therefore different contraction lengths of 0.07m and 0.039m (these are hereafter designated as LU48 and LU60 nozzles, respectively) The use of two nozzle sizes allowed for the influence of nozzle scale to be investigated, in particular jet Reynolds number. The nozzles were manufactured from 316L stainless steel to allow high temperature testing without distortion and were polished to produce a hydraulically smooth surface. The nozzle lips featured a 45° chamfer, which was included to reduce the nozzle lip thickness to 1mm. This improved optical access to the near nozzle exit lip region and reduced the distance between the measurement volume and the nozzle exit.

A further two nozzles whose nozzle exit diameters were 0.048m and 0.06m (designated LU48(P) and LU60(P), shown in Figure 2.6) were also investigated. Unlike the previous nozzles, these nozzles included an extension piece creating a short internal parallel wall section at the nozzle exit beyond the contraction, 0.0341m and 0.0318m in length for the LU48(P) and LU60(P) nozzles, respectively. The geometries of the contraction regions are identical to the LU48 and LU60 nozzles, described above. Nozzles with internal parallel wall exits have been shown to remove the presence of a vena-contracta [102] and the thin lip reduces the complexity of mesh generation for numerical simulations. For the current work, such nozzles were used to investigate the effect of the relaxation of the favourable pressure gradient on the nozzle exit boundary layer.

Due to manufacturing techniques, it was not possible to machine a sharp corner at the point at which the internal nozzle passage begins to contract, it is estimated that the transition consists of a small radius of approximately 5mm.
2.2 Pneumatic Probe Measurements

Pitot pressure surveys of the boundary layers at inlet and exit of the nozzles used pneumatic probes. Inlet and exit measurements involved different techniques and are discussed separately. Equipment which is common to both nozzle inlet and exit measurements is described first.

2.2.1 Common Instrumentation

2.2.1.1 Boundary Layer Pitot Probes

Boundary layer Pitot pressure surveys at nozzle exit and upstream of the nozzle inlet were conducted using the same boundary layer Pitot probe. The probe used was a United Sensors BR-025-12-C-11-120 boundary layer probe [103] with a flattened measuring tip to reduce measurement errors (Figure 2.7). The thickness of the the probe tip is 0.3175mm and the height of the sensing opening is 0.11mm. The probe features a short measuring head of length 2.54mm, which allows the probe to pass through a small access hole for measurements inside the delivery pipe. In order to deduce the boundary layer velocity profiles, a static pressure measurement was required. Different approaches were used for nozzle inlet and exit measurements and are discussed below.

2.2.1.2 Pressure Transducers

Wall static pressure (for nozzle inlet measurements) and Pitot probe pressure readings were measured using Huba Control Type 691 absolute pressure transducers. In order to improve resolution and reduce the influence of electrical noise and thermal effects on the measurements (by ensuring the magnitude of the transducer signal was several orders of magnitude larger than the noise) pressure transducers of two different pressure ranges were used. For low pressure flows (typically NPRs below 2) transducers with a range of 0 – 2.5 Bar were used. For higher NPRs transducers with a 0 – 6 Bar range were used. Where a wall static pressure tapping was available and dynamic pressures were low, a Huba Control type 652 1 Bar differential pressure transducer was used.

Prior to all tests all transducers were allowed to warm up for one hour in order to reach thermal equilibrium, after which the transducers were calibrated against a Druck DP610 0 – 7 Bar (gauge) pressure calibrator. These results were used to produce a third order polynomial calibration curve which was used to convert pressure transducer signal voltages to pressures. During prolonged test periods the calibration was periodically checked to identify any drift.
To reduce electrical noise further, a National Instruments SCXI 1000 signal conditioner chassis and SCXI-1302 terminal block and shielded cable were used to transfer signals from the transducers in the test cell to the data acquisition computer located in the adjacent control room. The distance between the static tappings and probes and their pressure transducers was minimised to reduce response times.

As nozzle operating conditions are usually described in terms of non-dimensional nozzle pressure ratio, defined as the ratio of nozzle inlet total pressure to ambient static pressure, the local ambient static pressure is required. Ambient static pressure was measured by a Druck type DPI 141 resonance barometer located in the control room. It was assumed that variations in static ambient pressure between the test cell and the control room were negligible.

2.2.2 Nozzle Inlet Boundary Layer Measurement

The literature review in Chapter 1 has revealed the influence that the favourable static pressure gradient which exists within propulsion nozzles can exert on nozzle boundary layers. To investigate this phenomenon fully, a knowledge of the boundary layer prior to acceleration is necessary in order to understand the extent of the change that takes place as the boundary layer negotiates the acceleration. At the outset of the project no access to the delivery duct upstream of the nozzle inlet was available, so a system was implemented in order to perform internal Pitot probe measurements.

2.2.2.1 Description

Access to the flow upstream was facilitated by the addition of a small section (measurement extension duct) between the end of the nozzle delivery pipe and the nozzle itself (Figure 2.8). The use of a removable section meant that no modifications to the existing duct or nozzles were required, and when inlet measurements were not needed it could be removed, reducing any flow disturbances which may otherwise have been caused by the probe access ports.

The extension comprises of a 0.07m long duct with identical internal diameter to the nozzle inlet, held in place by grub screws in the same manner as the nozzles. Two access ports were added at the same axial location, midway along the extension duct, 90° apart in the azimuthal direction to minimise mutual interference. One of these ports was 2mm in diameter and provided a wall static pressure tapping, the other was 3.10mm in diameter to provide a clearance fit for the boundary layer Pitot probe.

As the static pressures within the duct were several Bar greater than the ambient pressure
in the test cell, a sealing assembly was included to stop leakage around the probe which would interfere with pressure measurements. The sealing assembly comprised a gland packed with graphite yarn which acts as a seal capable of resisting high temperatures and pressures whilst minimising probe friction and was incorporated into the traverse (Figure 2.9).

2.2.2.2 Traversing and Data Acquisition System

Probe positioning and movement were achieved using a dedicated traverse system, mounted directly onto the extension duct (Figure 2.9). This has the advantage that it is not affected by any movement or thermal expansion or contraction of the duct as it is free to rotate and translate with the duct.

The probe was moved by an AIRPAX stepper motor and lead screw which provides a positional resolution of $0.0127\text{mm}$ (using half steps). Positional accuracy is dictated by the backlash of the lead screw which is quoted as $\pm 1$ step ($\pm 0.0254\text{mm}$). The stepper motor is controlled by a dedicated motor controller which can be positioned manually using a potentiometer to set the location as a percentage of the range of its travel or using a computer. As the traverse controller did not include a feedback mechanism, a calibrated linear potentiometer was also included in the design to provide an independent position of the probe. The potentiometer provided an absolute position of the probe so it could also be used to reposition the probe if its position were lost by the software (which would occur once the computer was switched off, for example, if tests were conducted on different days). This removed the need to reposition the probe manually, which would require dismounting the traverse assembly from the rig as there was no access to the probe once in position on the rig. Independent tests showed that no steps were missed provided that the stepping pulse frequency did not exceed 500Hz.

Control of the traverse and data acquisition was through a National Instruments PCI 6023e, 16 channel, 12 bit data acquisition card (with a maximum sample rate of 200000 samples per second) and a program written in LabVIEW 7.1. The LabVIEW program recorded the air supply total pressure, the pressure difference between the Pitot probe and wall static pressures, wall static pressure and the position of the probe based on the linear potentiometer. To improve the resolution of the dynamic pressure measurements at low nozzle pressure ratios, the range of voltages digitized by the DAQ was reduced using a controller within the software to reflect the low input signal voltage from the differential pressure transducer. Reducing the voltage range improved the resolution from approximately $50\text{Pa}$ to $12.5\text{Pa}$.
2.2 Pneumatic Probe Measurements

2.2.2.3 Test Procedure

Since there was no optical access to the probe when mounted on the rig, alignment of the probe was conducted prior to testing. Once aligned, the probe could be repositioned remotely (using the LabVIEW program) based on the position of the probe measured by the linear potentiometer. Measurements followed a predefined set of points, either generated by the LabVIEW software or imported from a coordinates file stored separately. Resolution in the near wall region was typically 0.1mm where wall normal total pressure gradients were large and increased to 1.0mm toward the centreine as the radial gradient tended toward zero.

A typical measurement sequence started at the wall location and comprised of moving the probe to the test desired location, followed by a 1 second pause to allow the probe pressure to equalise (based on probe response investigations of Trumper [62]). Transducer signals were then sampled for a further 2 seconds at an arbitrary sample rate of 2kHz. The sample duration was found to yield results independent of sample duration. A total of 92 spatial locations were typically measured in each traverse which had a duration of approximately 4.5 minutes, within the limited run-time experienced when operating in ‘blow-down’ mode.

2.2.3 Nozzle Exit Boundary Layer Measurement

Two independent measurement techniques were employed, a Pitot probe and a 2-component LDA system. The following discussion focuses on the probe measurements, the LDA measurements are described in Section 2.3.

2.2.3.1 Overview of Nozzle Exit Boundary Layer Measurements

Due to the long length of the primary nozzle delivery system, the boundary layer which forms within the delivery pipe has become very thick (approximately 20 - 25mm, see Figure 2.10) by the point at which the nozzle is attached. As a result, the majority of the fluid at the nozzle inlet has been contained within a boundary layer prior to acceleration and, therefore, will have a total pressure lower than centreline values due to losses caused by viscous effects (illustrated in Figure 2.10). At the nozzle exit, a similar proportion of the fluid will have a total pressure lower than the centreline value (as shown in Figure 2.11), with total pressures continuing to increase beyond the edge of the boundary layer (taken as the viscous effected region of the flow close to the wall in this context) which is significantly thinner at the nozzle exit (compared to the inlet) due to the effects of the acceleration (< 1mm). As a result it was necessary to traverse across the whole of the nozzle exit radius. Due to the large dynamic pressures at the exit, the boundary layer probe was not
able to withstand the aerodynamic loads when traversed into the main flow. To resolve this problem a dual Pitot probe was introduced, comprising of the boundary layer probe for performing the high resolution measurements in the near wall region where gradients were large, and a simple, reinforced, unflattened Pitot probe (inner diameter 1.39\text{mm}, outer diameter 1.65\text{mm}) to survey the rest of the jet exit where gradients were smaller (Figure 2.12) The Pitot probe head was carefully manufactured to ensure that the location of the tip was at the same axial position as the tip of the boundary layer probe. The two probes were connected to a pair of identical pressure transducers. Combining the two probes in the same probe holder allowed simultaneous measurement, removing any problems associated with matching operating conditions if two separate probes had been used in independent tests.

Since the fluid in the wall region at the nozzle exit was expected to be moving parallel to the nozzle wall, (which is inclined at up to 11° depending the nozzle geometry) relative to the centreline) an articulated probe support was used to allow the boundary layer probe to be parallel to the wall to minimise errors associated with probe incidence (Figure 2.13). As the flow at the centre of the jet is parallel to the axial direction, the Pitot probe was aligned with the axial direction. The centre of the Pitot probe used to perform the survey of the main flow was located one half of the exit diameter away from the tip of the boundary layer probe, such that when the boundary layer probe was at the wall the Pitot probe was on the centreline. Several Pitot probes were required depending on the contraction angle and exit diameter of the nozzle.

2.2.3.2 Axial Location of Nozzle Exit Boundary Layer Measurements

For data reduction purposes the value of the static pressure in the boundary layer is required to deduce the boundary layer velocity profiles from the measured Pitot pressure profiles. Unlike the nozzle inlet measurements, a wall static tapping at nozzle exit was not practical from manufacturing considerations and due to the large axial variations in static pressure that exist at the nozzle exit. For this reason, nozzle exit boundary layer measurements were made as close to the nozzle exit as possible but still displaced a small axial distance downstream (≈ 0.05mm, established by positioning the probe in contact with the nozzle then moving downstream by 0.05mm). This allowed a pressure at the wall location to be measured and this was taken as the static pressure of the boundary layer, assumed invariant across the boundary layer. Measuring outside of the nozzle greatly simplified the process of determining the wall location (discussed below in Section 2.2.3.4) but limited the range of experimental conditions to sub-critical pressure ratios, beyond which the nozzle will choke and the static pressure field outside the nozzle is no longer ambient static.
2.2 Pneumatic Probe Measurements

Since the measurements are being made just outside the nozzle, the question is raised, *to what extent do the measurements capture the boundary layer at the nozzle exit itself?*, since it is the initial stage of the free shear layer emanating from the nozzle lip that is actually being measured. This precise question was addressed by the study of Morris and Foss [104] on turbulent boundary layer to free shear layer transition and in Hamelin and Alving’s [105] study of low shear boundary layers.

In the work of Morris and Foss [104], measurements of an incompressible, two-dimensional turbulent boundary layer with Reynolds number $Re_\theta = 4650$ separating from a sharp edge showed that the outer region of the free shear layer downstream of the edge remains statistically identical to the original upstream boundary layer for a distance of several integral length scales. Morris and Foss [104] reported that measured profiles of mean velocity showed streamwise invariance for $y/\theta_\theta \geq 2$ for axial distances between $0 \leq x/\theta_\theta \leq 29$ where the subscript '0' denotes the values at the point of separation (Figure 2.14). Profiles of the RMS velocity showed streamwise invariance for $y/\theta_\theta \geq 4$ for axial distances between $0 \leq x/\theta_\theta \leq 29$ (Figure 2.15). Close inspection of the velocity profiles in Figure 2.14 reveals that the agreement is actually better than their published ranges when very close to the point of separation, with close agreement between velocity profiles for $y/\theta_\theta \geq 0.2$ for $0 \leq x/\theta_\theta \leq 3.5$.

The work of Hamelin and Alving[105], where the wall shear of a $Re_\theta = 2800$ boundary layer was removed by passing the boundary layer over a moving surface, showed similar results with streamwise invariance of the mean velocity for $y/\theta_\theta \geq 2$ for axial distances between $0 \leq x/\theta_\theta \leq 34$ and RMS velocity invariance above $y/\theta_\theta \geq 4$ extending between $0 \leq x/\theta_\theta \leq 30$. Based on preliminary nozzle exit boundary layer measurements, the measurement location used in the current work corresponds to an axial distance of $x/\theta \approx 2$, hence, based on the findings of Morris and Foss and Hamelin and Alving, it is expected that the measurements collected at the above axial location do indeed represent the nozzle exit conditions.

2.2.3.3 Traversing and Data Acquisition System

Probe positioning and traversing was achieved using a three-axis DANTEC lightweight 41T333 traverse, with a resolution of 6.25$\mu$m in all axes. Traverse control and data acquisition were managed through DANTEC miniCTA hotwire software modified to allow probe-based measurements. The same data acquisition card, signal conditioner and cabling were used as for the inlet measurements (Section 2.2.2.2).
2.3 Laser Doppler Anemometry

2.2.3.4 Test Procedure

Before testing began, the dual probe was positioned such that the tip of the boundary layer probe was located 0.05mm downstream of the nozzle lip and approximately 0.05mm below the edge of the lip so that a few points were measured beneath the inner nozzle wall location in order to allow better identification of the reference static pressure as discussed above. Axial location of the probe tip was achieved using a simple indicator which comprised of a small lamp and electrical circuit which was closed once the probe tip made contact with the nozzle. This allowed the probe to be repositioned, if necessary (to account for any movement of the probe or delivery pipe) from within the control room.

Unlike nozzle inlet measurements, nozzle exit boundary layer traverses were normal to the centreline and not normal to the nozzle wall. This may appear to be unconventional as boundary layers are usually considered in terms of velocity variation normal to the wall. However, where the nozzle is converging at the exit, traversing normal to the wall would involve movement inside the nozzle. Due to the rapid acceleration of the flow toward the nozzle exit and limitations of traverse positional resolution, any movement in the upstream direction would result in misleading velocity variations in the measured profiles.

Following a predefined set of measurement points, the dual probe was first moved toward the centre of the nozzle in 0.0125mm increments for approximately 0.8mm, performing detailed near-wall measurements taken by the boundary Pitot probe readings, then moved back to the start position. The dual probe was then moved from the centreline toward the wall in 1/40D increments and Pitot probe readings were taken to capture profile measurement over the whole of the nozzle exit. Between traversing to consecutive measuring points, there was a one second delay for the probe pressure to equalise, followed by a two second sampling period at a frequency of 2kHz, identical to the sampling procedure at the nozzle inlet. During a typical measurement run, 100 points were measured, 80 of which were within the near wall region. The typical duration of a run was 5 minutes. Variations in total pressure were typically less than ±1% during the test.

2.3 Laser Doppler Anemometry

The use of intrusive, pneumatic probes to survey fluid flows provides a simple method of quantifying the mean pressure field. The mean velocity field can be derived from measured total and static pressures as long as the total temperature is known. Unfortunately the data gathered are somewhat limited in scope (mean values only) and the range of operating conditions in which pneumatic probes may be confidently employed (subsonic NPRs).
2.3 Laser Doppler Anemometry

The review of literature on accelerated turbulent boundary layers has shown that mean profiles alone are not sufficient to establish whether a boundary layer has undergone relaminarisation, since departure from the familiar equilibrium velocity profile simply denotes a non-equilibrium turbulent boundary layer (which would be expected due to pressure gradient effects) but does not capture the extent of reversion to the laminar state [71]. The literature does, however, show that relaminarisation is accompanied by a large reduction in normalised turbulence level and the suppression of the Reynolds shear stresses, a result of an inability to respond as rapidly as the mean velocity to the change in static pressure. Information on the behaviour of the Reynolds stresses provides a much better means of identifying a relaminarising boundary layers. Boundary layer turbulence statistics are therefore desirable.

The use of Laser Doppler Anemometry (LDA) enables the capture of turbulence statistics. LDA is capable of measuring all types of flows, for example flows with high levels of turbulence and recirculation regions (where components of the velocities may be negative). The technique has been demonstrated in high speed flows by Lau et al.[106] and Jiang and Sisliam[107].

Despite the numerous advantages of LDA over intrusive measurement techniques, it is not without limitations, in particular, those which stem from the requirement to add tracer particles to the flow; these are discussed in the section on errors (Section 2.5) below. Nevertheless, it was decided that the use of LDA in the present experiments offered significant advantages. The 2-component LDA system and measurement techniques used to perform nozzle exit boundary layer measurements are described below. Some preliminary commissioning results for the LDA system in high speed flows is also provided. For details on the principles and practice of Laser Doppler Anemometry, the reader is directed to the works of Durst et al. [108], Buchhave et al. [109] and Albrecht et al. [110]; in the following section only matters of specific relevance to the present experiments are explained.

2.3.1 LDA System: Configuration and Operation

LDA measurements were conducted using a 2-component system which is located within the control room. The system comprises of a Spectra Physics Stabilite 1017 Argon-Ion 5w laser mounted on an optical rail in line with a DANTEC Fiberflow 60X41 transmitter. The transmitter contains a beam splitter which divides the light source into blue and green wavelengths (488nm and 514.5nm respectively) which are used for performing the measurements and a violet wavelength (476.5nm which is used as a reference for alignment purposes). The transmitter also contains a Bragg cell, an opto-acoustic device which adds a 40MHz frequency shift to one beam of the blue and green pairs of beams, this is required to remove directional ambiguity.
2.3 Laser Doppler Anemometry

Each of the beams are launched into fiberoptic cables using a series of fibre manipulators which allow the ‘tuning’ of the fibres which convey the beams to a DANTEC 85mm Fiberflow variable beam separation probe with a focal length of 310mm. Details of the measurement volume created at the crossing intersection of the beam pairs are shown below in Table 2.1. Measurements were conducted in ‘back-scatter’ mode where the transmitter probe also contains the collecting optics. Using the ‘back-scatter’ technique greatly simplifies the measurement process since a separate collecting optic is not required which would be difficult to align and would be affected by noise and vibration caused by the jet. The probe uses a multimode optical fibre to transmit the collected light which has been scattered by the seeding particles passing through the measurement volume to the colour separator and two separate photomultipliers.

Due to the small transverse thickness of the nozzle exit boundary layers, beam expanders were also used to improve measurement resolution and to help identify sources of errors associated with the size of the measurement volumes in flows with large velocity gradients and curvature (discussed in detail in Section 2.5). The DANTEC 55X12 beam expanders increase the diameter of the beams emitted by the probe by a factor of 1.98 which then reduces the width and length of the measurement volume by the same factor. The use of beam expanders also improves the Doppler signal as the power density is increased when the beams are focused on a smaller area, increasing data rates. Details of the measurement volume characteristics for the probe with beam expanders are shown in Table 2.2. The beam spacing is the distance between the shifted and unshifted beam before entering the front lens or beam expander if used. The fringe spacing is the distance between consecutive interference fringes within the measurement volume.

<table>
<thead>
<tr>
<th></th>
<th>$U_{BSA1}$</th>
<th>$U_{BSA2}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Wavelength (nm)</td>
<td>514.5</td>
<td>488</td>
</tr>
<tr>
<td>Focal Length (mm)</td>
<td>310</td>
<td>310</td>
</tr>
<tr>
<td>Beam Spacing (mm)</td>
<td>38</td>
<td>38</td>
</tr>
<tr>
<td>Fringe Spacing (μm)</td>
<td>4.205</td>
<td>3.989</td>
</tr>
<tr>
<td>Number of Fringes</td>
<td>35</td>
<td>35</td>
</tr>
<tr>
<td>Measurement Volume Diameter (mm)</td>
<td>0.151</td>
<td>0.143</td>
</tr>
<tr>
<td>Measurement Volume Length (mm)</td>
<td>2.549</td>
<td>3.332</td>
</tr>
</tbody>
</table>

Table 2.1: Measurement Volume Characteristics
2.3 Laser Doppler Anemometry

<table>
<thead>
<tr>
<th></th>
<th>$U_{BSA1}$</th>
<th>$U_{BSA2}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Wavelength (nm)</td>
<td>514.5</td>
<td>488</td>
</tr>
<tr>
<td>Focal Length (mm)</td>
<td>310</td>
<td>310</td>
</tr>
<tr>
<td>Beam Spacing (mm)</td>
<td>20</td>
<td>20</td>
</tr>
<tr>
<td>Expansion Ratio</td>
<td>1.98</td>
<td>1.98</td>
</tr>
<tr>
<td>Fringe Spacing (µm)</td>
<td>4.036</td>
<td>3.838</td>
</tr>
<tr>
<td>Number of Fringes</td>
<td>18</td>
<td>18</td>
</tr>
<tr>
<td>Measurement Volume Diameter (mm)</td>
<td>0.076</td>
<td>0.072</td>
</tr>
<tr>
<td>Measurement Volume Length (mm)</td>
<td>1.192</td>
<td>1.131</td>
</tr>
</tbody>
</table>

Table 2.2: Measurement Volume Characteristics With Beam Expanders

2.3.1.1 Signal Processing

The Doppler signals produced by the seeding particles passing through the measurement volume are processed by a dual-channel DANTEC F80 Burst Spectrum Analyser (BSA). The BSAs are controlled remotely from the control room computer via a standard Ethernet adapter, to convert the Doppler signals into velocities. For the present LDA equipment, the BSAs have a maximum Doppler frequency of 180MHz and a bandwidth of 120MHz to allow the measurement of high speed and highly turbulent flows.

The first stage of the signal analysis involves digitising the voltage time history of the photo-multiplier’s capture of the Doppler bursts, sampling the signal at regular time intervals. This is done automatically by the BSA, centreing the sampling period about the burst maximum then adjusting the sampling time based on the duration of each burst [111]. The digitised signal is then converted from the time domain to the frequency domain by a Digital Signal Processor (DSP) via an 8-bit Discrete Fourier Transform (DFT). The number of frequencies resolved by the DFT is equal to the number of samples, $N$. The frequency spacing (bin width) of the spectrum produced by the DFT is given by

$$\Delta f_s = \frac{1}{N\Delta t_s} = \frac{f_s}{N}$$

(2.1)

where $f_s$ is the sample frequency, $N$ is the total number of samples and $\Delta t_s$ is the period between samples. To improve resolution and accuracy, the number of samples is increased through the process of ‘zero padding’, i.e. adding a string of 0s to the end of the dataset processed by the DFT, decreasing the frequency spacing ($\Delta f_s$) without adding any additional spectral content to the output. The Doppler frequency, $f_d$, is derived and validated by curve fitting the frequency spectrum, and comparing the power spectral density of the two largest peaks. If the ratios of the two peaks is below a user-defined value, the burst is rejected,
otherwise the data are recorded for further processing.

2.3.1.2 Flow Seeding

As mentioned above, the LDA technique relies on the presence of seeding particles within the flow under investigation to reflect the light that forms the Doppler burst. For the boundary layer investigations and the subsonic jet plume measurements liquid droplet seeding was used. The seeding was produced by a TOPAS A M 230/DL aerosol generator (right hand side of Figure 2.16), powered by an independent air supply set and monitored from within the test cell control room. DEHS (Di(2-ethylhexyl)sebacate) was used as the seeding fluid. The seeder unit produces an aerosol with a modal droplet diameter of 0.25μm. The number of seeding particles was controlled by altering the air pressure supplied to the seeder. Optimum seeding, based on measured data rates, was produced when the seeder supply pressure was 0.5 Bar greater than the nozzle supply total pressure. The liquid droplet seeding was added to the high pressure air using an injector located downstream of the combustor, approximately 2m upstream of the nozzle exit, to allow the particles to become uniformly distributed through the flow which was indicated by constant data rates obtained within the core region of the jet. For the under-expanded jet plume measurements it was not possible to use liquid droplet seeding. This was due to condensation of the moisture in the entrained ambient air within the shear layers (where the static temperatures were very low due to high velocities and a total temperature equal to the ambient temperature) causing very poor signal to noise ratios. Instead, a solid particle seeded system was used. This system comprised a SCITEK Consultants PS – 10, solid particle seeder for high pressure flows and is shown on the left hand side of Figure 2.16. The seeding particles were contained within a rotating canister in powder form; this deposited the powder in front of a series of high pressure microjets which were responsible for fluidising the powder. The seeding density is set and monitored remotely in the test cell control room by adjusting the rotational speed of the canister. Struers AP-D alumina polishing powder with 0.3μm diameter particles was used as the seeding. The powder was kiln dried before use to stop the powder caking in the seeder unit and in the main flow. The air supply for the microjets was taken from the rig air supply, upstream of the globe valve in the test cell. Seeding was introduced into the primary nozzle supply duct at the same location as the liquid droplet seeding.

2.3.1.3 Traversing and Data Acquisition System

LDA probe positioning and traversing (Figure 2.17) were achieved using the same three-axis DANTEC lightweight traverse used for the nozzle exit Pitot probe measurements and described in Section 2.2.3.3. Traverse control and data acquisition were managed through
2.3 Laser Doppler Anemometry

DANTEC BSA Flow software. The software provides full control of measurement parameters (such as centre frequency and bandwidth). Further details are available in [112].

2.3.1.4 Alignment

When performing LDA measurements, correlations such as Reynolds shear stresses require the velocity samples from the two channels (blue and green) to be coincident and simultaneous. To ensure that the measured velocities from different channels are from the same particle at the same time, a coincidence window (maximum time interval between signals being received from different channels) is introduced, either through the hardware (BSA) or through the controlling software [112]. If the signals from the two channels arrive with a time difference greater than the coincidence window, the signals were deemed to be non-coincident and are excluded from the measurements. The coincidence window was set as the time taken for a particle traveling at the maximum velocity expected in the flow to cross the measurement volume [112]. The maximum speed was taken as $360\text{m/s}$ (corresponding to the velocity of fully-expanded flow at an NPR of 2.4 and a total temperature of $288K$) resulting in a coincidence window of $0.14\mu s$. It was important to ensure that the two pairs of beams intersect properly at the same location, otherwise individual particles may not cross the intersection of both the blue and green beams, resulting in poor data rates. The problem becomes more acute as the size of the intersection volumes of the two sets of beams becomes smaller, as is the case when beam expanders are used. To ensure that the location of the beam intersections for the two beam pairs occurred at the same position, the intersection was checked by passing the beams through an objective pinhole, mounted on the optical rail which supports the probe, and located at the intersection of the beams, then projecting the beams on to the the wall of the test cell. If necessary, the intersection was adjusted using the beam adjusters located on the probe.

The optical axes of the beam pairs emitted by the transmitting probe were orientated at $\pm 45^\circ$ to the jet centreline such that both of the channels would be measuring velocities of similar magnitude, which allowed identical centre frequencies and bandwidths to be used. This orientation also allowed the measurement volume to be located closer to the nozzle lip compared to the case where the two beam pairs were normal and aligned with the centreline. The correct orientation of the transmitter probe was achieved by projecting the beams onto the wall of the test cell and rotating the probe until the beams coincided with a datum marked on the wall. A schematic of the orientation of the beam pairs relative to the nozzle centreline is shown in Figure 2.18.
2.3 Laser Doppler Anemometry

2.3.1.5 Test Procedure

In order to maximise the data rates attained by the LDA, prior to all testing the laser beams were ‘tuned’. This involved aligning the end of each of the transmission fibres using the fibre manipulators to ensure that the beam was correctly launched into the fibre. Alignment was gauged against the power output of each of the beams, as measured by a laser power meter, and adjusted until the maximum was obtained. During prolonged test periods (more than 2 hours) the alignment was checked periodically.

The measurement volume was positioned a small axial distance downstream of the nozzle exit in a similar manner to the nozzle exit probe measurements discussed in Section 2.2.3.2. The location was dictated by the need for unobstructed beam paths between the probe and the measurement location across the whole of the boundary layer. The final location of the centre of the control volume corresponded to an axial distance of approximately 15 nozzle exit boundary layer momentum thicknesses downstream of the nozzle lip for LU60 measurements with the use of beam expanders. Based on the observations of Morris and Foss [104] and Hamelin and Alving [105] this location of the measurement volume is within the range where the measured profiles are still representative of the nozzle exit boundary layer (Section 2.2.3.2).

Following a predefined set of measurement points, the probe was moved toward the centreline in 0.0125mm increments. At each measurement point 10000 samples were taken. Where measurements were very close to the nozzle wall or in front of the nozzle lip, data rates were low (typically a few hundred Hz compared to 10kHz at the edge of the boundary layer). At these points it was not practical to wait and record 10000 samples due to operating time constraints when operating in ‘blow-down’ mode. For this reason a time constraint was added. After a maximum sample period of 10 seconds, the traverse moved to the next measurement point regardless of whether 10000 samples had been collected or not. The lowest number of samples for measurement locations near the wall was typically around 2000. The effect of the number of samples on the time-averaged velocity statistics is discussed in section 2.5.2.1

The duration of a run to measure a single profile varied depending on measurement data rates, but was typically of 5-6 minutes duration.
2.4 Data Reduction

2.4.1 Pneumatic Probe Measurements

Velocity profiles for the boundary layers measured using Pitot probes were derived using simple gas dynamics. A relationship between the local total to static pressure ratio and Mach number can be derived such that:

\[
\frac{P}{\rho} = \left(1 + \left(\frac{\gamma - 1}{2}\right)M^2\right)^{\frac{\gamma}{\gamma - 1}} \tag{2.2}
\]

which can be used to form an expression for the local Mach number:

\[
M = \left\{\left\{\left(\frac{P}{\rho}\right)^\frac{\gamma - 1}{\gamma} - 1\right)\left(\frac{2}{\gamma - 1}\right)\right\}^{\frac{1}{2}} \tag{2.3}
\]

The static pressure used was the measured wall static value for the nozzle inlet profiles and the static pressure deduced from the measured Pitot profiles for the nozzle exit profiles (described below in Section 2.4.3.2). The variation of static temperature across the boundary layer was not measured directly but was derived from the Mach number distribution, deduced from the measured pressures and the (assumed constant) total temperature of the flow measured within the nozzle supply duct. It was assumed that the walls of the nozzle and delivery duct were adiabatic. Therefore, the total temperature at any point within the boundary layer is constant and equal to that of the free-stream, the wall temperature is assumed equal to the free-stream total temperature (T), implying that a recovery factor (r) of unity is assumed. The static temperature (t) at any point in the boundary layer with velocity U is a quadratic function of the Mach number [66]:

\[
\frac{T}{t} = 1 + \frac{\gamma - 1}{2}M^2 \tag{2.4}
\]

Which can be written as a function of the local Mach number as:

\[
t = \frac{T}{1 + \frac{\gamma - 1}{2}M^2} \tag{2.5}
\]

This permits the local speed of sound to be deduced:

\[
a = (\gamma R t)^{\frac{1}{2}} \tag{2.6}
\]

Where R is the universal gas constant (287 J/kgK) and γ is the ratio of specific heats (taken as 1.4 for the current investigations), which can then be used to define the local velocity:

\[
U = aM \tag{2.7}
\]
2.4 Data Reduction

Where variation of density was required, it was calculated using the equation of state:

\[ \rho = \frac{p}{Rt} \]  

(2.8)

### 2.4.2 LDA Measurements

The velocity components measured by the two LDA channels were first filtered to remove any spurious samples that occurred due to the large bandwidth required to measure the whole of the boundary layer in a single operation. Before deriving time-averaged statistical quantities such as mean velocities and Reynolds stresses, the measured instantaneous velocity components \( U_{BSA1} \) and \( U_{BSA2} \) were transformed into Cartesian components. As the orientation of each pair of beams relative to the nozzle is known (Section 2.3.1.5). The instantaneous Cartesian velocity components \( U_{inst} \) (is in the \( x \) direction and \( V_{inst} \) is in the \( y \) direction) can be defined from:

\[ U_{inst} = U_{BSA1} \cos \theta - U_{BSA2} \sin \theta \]  

(2.9)

\[ V_{inst} = U_{BSA1} \sin \theta + U_{BSA2} \cos \theta \]  

(2.10)

Where \( \theta \) is the orientation of the beams relative to the nozzle centreline. Solving the above equations allows the formation of an optical transformation:

\[
\begin{bmatrix}
U_{inst} \\
V_{inst}
\end{bmatrix} =
\begin{bmatrix}
c_{11} & c_{12} \\
c_{21} & c_{22}
\end{bmatrix}
\begin{bmatrix}
U_{BSA1} \\
U_{BSA2}
\end{bmatrix}
\]

(2.11)

where

\[
\begin{bmatrix}
c_{11} & c_{12} \\
c_{21} & c_{22}
\end{bmatrix} =
\begin{bmatrix}
\cos \theta & -\sin \theta \\
\sin \theta & \cos \theta
\end{bmatrix}
\]

(2.12)

which simplifies to:

\[
\begin{bmatrix}
\frac{\sqrt{2}}{2} & -\frac{\sqrt{2}}{2} \\
\frac{\sqrt{2}}{2} & \frac{\sqrt{2}}{2}
\end{bmatrix}
\]

(2.13)

for the current investigation where \( \theta = 45^\circ \).
2.4 Data Reduction

From the instantaneous velocity components time-averaged quantities can be defined. The mean velocity is defined as:

\[ U = \sum_{i=1}^{n} \eta_i U_{\text{inst},i} \]  

(2.14)

where \( \eta_i \) is a weighted factor and \( n \) is the total number of samples taken. \( \eta_i \) represents the transit time weighting factor, defined as:

\[ \eta_i = \frac{t_i}{\sum_{i=1}^{n} t_j} \]  

(2.15)

where \( t \) is the transit time of the \( i^{th} \) seeding particle. Its use is discussed in more detail in Section 2.5.

The root mean square of the fluctuating velocity components is defined as the square root of the variance, which is defined as:

\[ \sigma_u^2 = \sum_{i=1}^{n} \eta_i (U_{\text{inst},i} - U)^2 \]  

(2.16)

\[ \sigma_v^2 = \sum_{i=1}^{n} \eta_i (V_{\text{inst},i} - V)^2 \]  

(2.17)

The cross-correlation or Reynolds shear stress, \( \overline{u'v'} \) is defined as:

\[ \overline{u'v'} = \sum_{i=1}^{n} \eta_i (U_{\text{inst},i} - U)(V_{\text{inst},i} - V) \]  

(2.18)

2.4.3 Derived Quantities

Over the wide range of operating conditions that can be achieved in the HPNTF high velocities occur in both the delivery duct upstream of the nozzle inlet and particularly at the nozzle exit where velocities reach the local speed of sound. Static temperature variations across the boundary layers can therefore be of sufficient magnitude that the influence of local fluid property variation within the boundary layer may be significant and cannot be ignored particularly for fluid density. The strength of this effect clearly varies as a function of Mach number.

The varying degree of these effects across the range of test conditions makes comparison of results more difficult, particularly when considering the evolution of the highly accelerated boundary layer through the nozzle between different nozzle geometries. Decoupling
the effects on the boundary layer of high and increasing Mach number as the flow accelerates through the nozzle and the influence of the highly favourable static pressure gradient is hence problematic. To analyse the boundary layer under compressible flow conditions, local static pressures and temperatures are required in order to derive the variation of fluid density across the boundary layer. Although this is possible for the profiles measured using pneumatic probes (albeit with additional sources of errors due to local static temperatures not being measured directly but rather deduced from Mach number distributions and assumed adiabatic walls and unity temperature recovery factors), it is not possible to deduce the density distributions for the profiles measured using the Laser Doppler Anemometry technique as only velocity measurements are made. Since it is desirable to be able to compare and contrast probe and LDA data for the same operating conditions, a suitable data processing approach is required to allow the two types of measurements to be compared.

2.4.3.1 Compressibility Correction

It is generally accepted that turbulent fluctuations of density in high speed flows have little influence on the turbulent structures provided that the root mean square of the density fluctuations is small compared to the mean density (Morkovin's Hypothesis [66]) such that:

\[
\frac{\sqrt{\rho^2}}{\bar{\rho}} \ll 1
\]  

(2.19)

Bradshaw [113] has pointed out that this implies that the turbulent structures of wake and boundary layer flows at freestream Mach numbers of less than 5 and jet flows at Mach numbers of less than 1.5 closely resemble those of a constant density flow. It is argued therefore that methods used to compute and analyse incompressible flows can be applied to compressible flows. The direct effect of compressibility on high Reynolds number, turbulent boundary layers over a wide range of constant free-stream Mach numbers was examined in detail in the study by Winter and Gaudet [114] and later by Gaudet [115]. Temperature and Pitot pressure measurements were made in a zero pressure gradient flow on the side wall of an 8ft x 8ft supersonic wind tunnel, over a free-stream Mach number range from 0.2 to 2.8 with Reynolds numbers, based on the development length of the boundary layer, ranging from 100 million (for supersonic conditions) to 200 million (at low subsonic conditions). Local skin friction was measured using a large 14.5 inch diameter skin friction balance. Their analysis was also extended to data gathered by other workers.

This investigation showed that if the boundary layers were analysed in a so-called kinematic form (where only velocities were considered, i.e. the density terms in the classical definitions of momentum and displacement thicknesses were ignored), then the characteristic
2.4 Data Reduction

Integral thicknesses were independent of Mach number over a wide range of Mach numbers. The authors conjectured that the same would be true up to Mach numbers of order 5 where the magnitude of the fluctuating components of velocity became supersonic.

Studies of the variation of the kinematic form of the shape factor $H_{12i}$, the ratio of the kinematic displacement and momentum thicknesses showed no significant dependency on Mach number, instead being solely dependent on the kinematic momentum thickness Reynolds number, defined as:

$$Re_{i} = \frac{\rho_{i} U \delta_{i}}{\mu}$$  \hspace{1cm} (2.20)

(note that the fluid properties $\rho$ and $\mu$ were evaluated at the free-stream edge of the boundary layer). The kinematic definition of boundary layer parameters was extended to the law of the wall and law of the wake velocity distributions suggested originally by Von Kármán and Coles. Winter and Gaudet [114] defined an effective friction velocity ($u_{*i}$) based on kinematic values (taken at the freestream):

$$u_{*i} = \frac{\tau_{w}}{\rho_{\infty} U_{\infty}^{2}}$$ \hspace{1cm} (2.21)

The kinematic form of the velocity profile, scaled in inner wall units becomes

$$\frac{U}{u_{*i}} = \frac{1}{\kappa} \ln \left( \frac{y}{\nu_{*i}} \right) + \frac{2\Pi}{\kappa} \left( \frac{y}{\delta} \right)$$ \hspace{1cm} (2.22)

Where the subscript $i$ denotes a kinematic quantity. The defect law in kinematic form becomes:

$$\frac{U_{\delta} - U}{u_{*i}} = \frac{1}{\kappa} \ln \left( \frac{y}{\delta} \right) - \frac{2\Pi}{\kappa} \left( \frac{y}{\delta} \right)$$ \hspace{1cm} (2.23)

Comparison of mean velocity profiles in inner wall scaling using kinematic values (at the edge of the boundary layer) for a wide range of Mach numbers (0.4 – 2.8) was shown in [114] to produce close agreement within the bounds of reasonable experimental error. The kinematic form of boundary layer parameters was previously used in the nozzle exit boundary layer studies of Lepikovsky [61] and Lepikovsky et al. [5], the study of relaminarisation by Nash-Webber and Oates [87] and in the review of compressible turbulent boundary layer by Fernholz and Finnley [68], and hence it will be used also in the present work.

2.4.3.2 Integral limits

In order to calculate any integral quantities or other derived parameters, the lower and upper limits of integration must be defined, namely the locations of the wall (lower limit) and the
edge of the boundary layer (δ, upper limit). Details of the boundary layer integrals for axisymmetric boundary layers are described in the following sections.

For nozzle inlet measurements, the edge of the boundary layer was defined as the point at which the local velocity was 99.5% of the centreline value.

The classical definition of the edge of the boundary layer corresponding to the point where the local velocity reaches 99% of the centreline value could not be employed for the nozzle exit boundary layers. This was due to total pressure deficits beyond the viscous influenced region (a legacy of the thickness of the boundary layer prior to the contraction) and the presence of a vena-contracta. Instead the edge of the boundary layer was arbitrarily defined as the point where the radial gradient of the mean velocity was less than 1% of its maximum value. For the LDA measurements of the nozzle exit boundary layer, the same definition was also used. As turbulence data were also available from the LDA measurements, the location of the boundary layer edge was also defined as the point where the turbulence levels decreased to free stream levels. Comparisons between the two methods of defining the location showed good agreement.

As discussed in section 2.2.3, nozzle exit boundary layer measurements were conducted a small distance downstream of the nozzle lip (approximately 2 and 15 times the exit boundary layer momentum thickness for the probe and LDA techniques respectively). As a result interpolation of the measured profile was required in order to define the effective location of the nozzle wall for the purposes of defining the lower limit of integration for the integral parameters. For the Pitot pressure measurements, the interpolation was achieved through linear extrapolation between the first few measured pressures at points clearly inside of the boundary layer and a few measured points which were clearly outside it. The difference between these point was very clear as shown in Figure 2.19. The effective wall location was defined as the point at which the two extrapolated lines intersected each other. In general, only one or two points were affected by the interpolation, corresponding to the point of inflection of the low speed side of the shear layer. For the LDA measurements, the effective wall location was established by interpolating the velocity profile to zero velocity, as shown in Figure 2.20.

2.4.3.3 Compressible Boundary Layer Integral Parameters

This section provides, for completeness, the formulae used to evaluate the integral parameters of the compressible boundary layers studied in this thesis.

The displacement thickness (δ*) of a boundary layer represents the defect in the mass flow rate within the boundary layer compared to the equivalent inviscid flow [64]. The
compressible form of the displacement thickness of an axisymmetric internal flow boundary layer is defined as:

\[ \delta^* = R - (R^2 - 2RI_1)^{\frac{1}{2}} \]  

(2.24)

where \( R \) is the internal radius of the pipe and \( I_1 \) is the integral defined as:

\[ I_1 = \int_0^\delta \left(1 - \frac{\rho U}{\rho_0 U_0}\right) \left(1 - \frac{y}{R}\right) dy \]  

(2.25)

Where the subscript \( \delta \) denotes quantities at the edge of the boundary layer, and the distance, \( y \), is evaluated from the wall and not the nozzle centreline.

The momentum thickness represents the defect of momentum flux within the boundary layer compared to the inviscid flow resulting from skin friction drag [64] and is defined as:

\[ \theta = R - (R^2 - 2RI_2)^{\frac{1}{2}} \]  

(2.26)

Where \( I_2 \) is the integral defined as

\[ I_2 = \int_0^\delta \frac{\rho U}{\rho_0 U_0} \left(1 - \frac{U}{U_0}\right) \left(1 - \frac{y}{R}\right) dy \]  

(2.27)

The boundary layer shape factor is defined as the ratio of the displacement and momentum thicknesses:

\[ H_{12} = \frac{\delta^*}{\theta} \]  

(2.28)

The momentum thickness Reynolds number is defined in terms of edge of boundary layer edge properties:

\[ Re_\theta = \frac{\rho_0 U_0 \theta}{\mu_0} \]  

(2.29)

2.4.3.4 Kinematic Boundary Layer Integral Parameters

The kinematic forms of the displacement and momentum thicknesses of an axisymmetric internal flow are defined as

\[ \delta_i^* = R - (R^2 - 2RI_{i1})^{\frac{1}{2}} \]  

(2.30)

\[ \theta_i = R - (R^2 - 2RI_{i2})^{\frac{1}{2}} \]  

(2.31)
Where \( I_{1i} \) and \( I_{2i} \) are integrals defined as:

\[
I_{1i} = \int_0^\delta \left( 1 - \frac{U}{U_\delta} \right) \left( 1 - \frac{y}{\delta} \right) dy \\
I_{2i} = \int_0^\delta \frac{U}{U_\delta} \left( 1 - \frac{U}{U_\delta} \right) \left( 1 - \frac{y}{\delta} \right) dy
\]

(2.32)  
(2.33)

The kinematic boundary layer shape factor is defined as:

\[
H_{12i} = \frac{\delta_i}{\delta_i}
\]

(2.34)

The kinematic momentum thickness Reynolds number is defined in a similar manner to the compressible form, instead using the kinematic momentum thickness:

\[
Re_\delta = \frac{\rho_\delta U_\delta \delta_i}{\mu_\delta}
\]

(2.35)

### 2.4.3.5 Skin Friction

The High Pressure Nozzle Test Facility does not permit direct measurement of skin friction. Therefore, a method of estimating the skin friction from the measured mean velocity profiles was employed. Due to the high Reynolds numbers within the delivery pipe the location of the first measurement point (typically 0.01mm away from the wall) is of sufficient distance away from the wall to be outside the laminar sub-layer \((0 \leq y^+ \leq 5)\) and buffer regions \((5 \leq y^+ \leq 25)\) and resides within the logarithmic region.

Local skin friction coefficients could therefore be derived from single point velocity measurements using the log-law. However, the results using this method displayed a degree of scatter resulting from small drifts in the supply total pressure, probe flexing and transducer noise. Points near the wall introduce additional uncertainties in terms of measurement location and probe displacement effects (Section 2.5). In order to reduce these errors, the multiple point Clauser plot method [116] was adopted.

If both sides of the kinematic form of the logarithmic Law of the Wall:

\[
\frac{U}{u_{ri}} = \frac{1}{\kappa} \ln \left( \frac{y}{\nu} \right) + B
\]

(2.36)  

(where \( \kappa \) is the Von Kármán constant and the additive constant \( B \) is derived experimentally) are multiplied by \( \frac{u_{ri}}{U_\delta} \) the result is the expression

\[
\frac{U}{U_\delta} = \frac{u_{ri}}{\kappa U_\delta} \ln \left( \frac{U_\delta y}{\nu_\delta} \right) + \frac{u_{ri}}{\kappa U_\delta} \ln \left( \frac{u_{ri}}{U_\delta} \right) + \frac{B u_{ri}}{U_\delta}
\]

(2.37)
2.5 Error Sources

Substituting the local kinematic skin friction coefficient:

\[ c_{fi} = \frac{2\tau_{wi}}{\rho S U_d^2} = 2 \left( \frac{\frac{\tau_{wi}}{U_d}}{U_d} \right)^2 \]  \hspace{1cm} (2.38)

into Equation 2.37 yields:

\[ \frac{U}{U_d} = \frac{1}{\kappa} \sqrt{\frac{C_f}{2} \ln \left( \frac{U_d y}{\nu_0} \right)} + \sqrt{\frac{C_f}{2} \left( B + \frac{1}{\kappa} \ln \sqrt{\frac{C_f}{2}} \right)} \]  \hspace{1cm} (2.39)

Equation 2.39 can then be used to generate a family of Clauser curves (straight lines on a \( \frac{U}{U_d} - \ln \frac{y}{\nu_0} \) chart) for varying values of \( c_{fi} \). The experimental data when plotted on the same chart will identify the appropriate value of the local skin friction coefficient. An example of a Clauser plot is shown in Figure 2.21.

2.5 Error Sources

During any experimental investigation it is inevitable that errors will be present in the results, such as statistical errors, measurement resolution, or the accuracy of related systems such as transducers. Since the total measurement error will be a combination of many factors, and these will vary with operating conditions, it is difficult to quantify errors accurately. However, it is possible to identify the main sources of error associated with the measurement of wall bounded flows at high speeds reported in this thesis. The following section presents these main error sources for pneumatic probe and LDA measurements. Where possible, attempts are made to quantify these errors.

2.5.1 Pressure Measurements

For pneumatic probe measurements in high speed flows the main contributing factors to experimental errors are Mach number effects, probe alignment, displacement effects and wall proximity effects.

In high speed flows, local shockwaves can form in front of the tip of the probe where the local acceleration of the flow caused by the displacement effect of the probe head causes the flow to become supersonic. The subsequent loss in total pressure due to the shockwave results in a decrease in the pressure measured by the probe [100]. United Sensors [103] state that the upper Mach number limit of Pitot pressure readings is M0.95, beyond which the formation of shockwaves will result in erratic pressure measurements with slight changes in
operating conditions or proximity to solid boundaries. Their results showed the error to be less than 2% for Mach numbers below 0.95.

If the head of the Pitot probe is not aligned with the direction of the flow errors in the measured pressure will result; the errors are typically small at small incidence, increasing rapidly at large incidence. The sensitivity to probe incidence is highly dependent on the shape of the probe tip, with 1% error ranges for incidences of \( \pm 11^\circ \) for square ended probes (as used in the present experiments) increasing to \( \approx \pm 30^\circ \) when the inside of the probe tip has a 15% chamfer. Ower \cite{100} also reports that the probe stem influences sensitivity.

In the current investigation it was necessary to align the probe with the wall for convergent nozzle exit boundary layer measurements, as discussed above. Since the boundary layer and standard Pitot probes were traversed across the exit plane, their incidence to the oncoming flow varies. The greatest variation is seen by the standard Pitot probe whose incidence increases from 0° at the centreline to \( +11^\circ \) at the wall. In light of this, the sensitivity to incidence (pitch) of both the boundary layer and standard Pitot probes used here were investigated using a calibration rig designed to calibrate 5-hole probes and described by Wray \cite{117}. Pressure measurements were made in the potential core of a low turbulence jet issuing from a smoothly contoured nozzle with a mean velocity of approximately 38\( m/s \). The sensitivity to incidence of the boundary layer probe and the two standard Pitot probes for exit surveys for the LU48 and LU60 nozzles to pitch is shown in Figure 2.22. The results show the probes are relatively insensitive to incidence, with variations in the measured dynamic pressure of approximately 0.2%, 0.25% and 0.25% for the boundary layer, LU48 and LU60 Pitot probes at an incidence of \( \pm 5^\circ \) respectively (compared to the pressure measured at zero incidence) rising to 1.5%, 0.75% and 1.0% at an incidence of \( \pm 10^\circ \). The influence of the stem is also demonstrated in Figure 2.22 by the asymmetry of the variation when the probes are pitched up and down. The influence of probe incidence in high speed flows is similar to incompressible flows even in supersonic conditions \cite{100} therefore it is assumed that these results are representative of the high speed flows in which the current probe measurements are taken.

Measurements in close proximity to solid boundaries are subject to wall effects, where the measured pressure is less than the actual pressure; in effect the aerodynamic centre of the probe is displaced toward the wall compared to the geometric centre. The study of Quarmby and Das \cite{118} for flattened probes showed a velocity error of between \(-1\%\) and \(-2\%\) when the probe is less than 2 times its thickness away from the wall. Generally, the errors associated with wall proximity effects in the present data are small. For the nozzle inlet measurements, the boundary layer thickness was approximately 70 times greater than the thickness of the probe and the region affected is negligible. For the nozzle exit boundary
layers it is not possible to quantify the influence of any wall effects as the measurements are taken outside of the nozzle and the probe is in close proximity to two surfaces, the nozzle lip and the nozzle internal wall.

Additional errors arise from the effects of velocity gradients across the mouth of the Pitot probe. The result is a displacement of the aerodynamic centre of the probe away from the geometric centre toward the region of higher velocity resulting in the velocity measured by the probe being greater than the actual velocity. The study of Quarmby and Das [118] on flattened tipped probes, similar to those used here, gives the displacement as:

$$\frac{\delta}{H} = 0.19$$

(2.40)

where $\delta$ is the displacement and $H$ is the thickness of the probe. This corresponds to a constant positional error of 0.06mm for the probes used in this investigation. The more recent work of McKeon et al. [119] proposed a displacement correction based on measured velocity gradients such that no correction is applied in regions of no velocity gradient. The displacement correction is defined as

$$\frac{\Delta U}{d} = 0.15 \tanh (4\sqrt{\alpha})$$

(2.41)

where $\alpha$ is the non-dimensional velocity gradient

$$\alpha = \frac{d}{2U(y_c)} \left. \frac{dU}{dy} \right|_c$$

(2.42)

and $y_c$ is the distance from the wall to the geometric centre of the probe.

For the nozzle exit boundary layer measurements where the probe thickness is significant compared to the thickness of the boundary layer, the latter correction technique was adopted.

### 2.5.2 LDA Measurements

Sources of error attributed to the Laser Doppler Anemometry technique can be broadly divided into two categories, statistical and systematic. Statistical errors arise from uncertainties when using a finite number of samples of particle velocities from which to derive time-averaged mean velocities and turbulence statistics; these are discussed in Section 2.5.2.1. Systematic errors arise from the manner in which the measurements are made, in particular the use of tracer particles and a finite measurement volume. The main sources of systematic errors in the current investigation are velocity bias (which results from varying particle arrival frequencies and is discussed in Section 2.5.2.2) and the effect of large, non-uniform
velocity gradients, referred to as velocity gradient broadening and this is described in Section 2.5.2.3. Details on additional biases are available in Durst et al. [108] and Edwards [120].

2.5.2.1 Statistical Errors

As data rates were lowest (less than 1kHz) in the near-wall region due to low seeding densities, it is important to establish the level of confidence which can be placed on the results in this region in particular. The degree of confidence can be estimated using methods available in standard texts on statistics assuming the velocity samples measured by the LDA are normally distributed about the mean and are all statistically independent. In the case of boundary layer measurements the assumption of a normal (Gaussian) distribution is not entirely accurate but such assumptions are reasonable for the purpose of estimating statistical errors. As the highest measured data rates are more than an order of magnitude lower than the eddy turn-over frequency in the boundary layer (the rate at which an eddy whose scale is equal to the thickness of the boundary would pass a fixed point) in the boundary layer, it is almost certain that all the particles sampled will be associated with different turbulent structures and, therefore, statistically independent.

For the mean of N samples of a normally-distributed random variable the confidence level can be determined from:

\[ \epsilon = \frac{c\sigma}{\sqrt{N}} \]  

(2.43)

where \( c \) is a coefficient corresponding to the chosen confidence level (1.960 for a confidence level of 95%, and 2.576 for a confidence level of 99%) and \( \sigma \) is the square root of the variance. Assuming that the mean is equal to \( U \) and the variance is equal to \( \overline{u'\overline{u'}} \), Equation 2.43 can be re-arranged to give the error for the chosen confidence level as:

\[ \frac{\epsilon}{U} = \frac{c\overline{u'}}{\sqrt{N}} \]  

(2.44)

At the edge of the boundary layer, where turbulence levels are typically 2 – 3% and 10000 samples were made, the measured mean velocity is expected to lie within ±0.05% of the true mean. In the near wall region where data rates are low and turbulence levels were around 20 – 25%, the uncertainty increases, with measured values of the mean lying within ±6% of the true mean.

The statistical uncertainties in the measured normal stresses can be estimated by determining the confidence interval of the variance of a normal distribution assuming that the measured normal shear stresses are representative of the variance. The method adopted follows the examples of Kreyszig [121].
Assuming a confidence level level of 99%, the confidence interval can be defined as

\[ k_2 \leq \sigma^2 \leq k_1 \]  \hspace{1cm} (2.45)

where

\[ k_1 = (N - 1) \frac{s^2}{c_1} \]  \hspace{1cm} (2.46)

and

\[ k_1 = (N - 1) \frac{s^2}{c_2} \]  \hspace{1cm} (2.47)

where \( s^2 \) is equal to the normal stress and the constants \( c_1 \) and \( c_2 \) are approximated as

\[ c_1 = \frac{1}{2} \left( \sqrt{(2m - 1)} - 2.33 \right)^2 \]  \hspace{1cm} (2.48)

\[ c_2 = \frac{1}{2} \left( \sqrt{(2m - 1)} + 2.33 \right)^2 \]  \hspace{1cm} (2.49)

where \( m \) is the number of degrees of freedom and is large \((m >> 100)\). In the near wall region the measured normal stresses are expected to lie within \( \pm 5\% \) of the true value.

Examples of statistical errors are shown in the measured mean and RMS axial velocity profiles shown in Figures 2.23 and 2.24.

### 2.5.2.2 Velocity Biasing

Velocity biasing occurs in all LDA systems when operating in 'burst' mode (or 'Individual Realization', IR mode) where each particle passing through the measurement volume is measured. For a flow of uniform seeding particle concentration, the likelihood of a particle traveling through the measurement volume at a velocity greater than the average is much higher than one traveling below the average speed. This is due to the greater volume of the higher speed fluid, therefore a greater number of seeding particles, passing through the measurement volume in a given time period, as first identified by McLaughlin and Tiederman [122]. The probability of a particle passing through the measurement volume is proportional to the fluid velocity [123]. This results in a bias in the probability density function toward higher velocities. McLaughlin and Tiederman quantified the bias in a uniformly seeded flow such that

\[ \langle U \rangle_m \approx \langle U \rangle \left( 1 + \frac{\sigma^2}{\langle U \rangle^2} \right) \]  \hspace{1cm} (2.50)
where the subscript ‘m’ denotes the measured value, and shows that the bias increases with the square of the turbulence intensity.

The problem becomes more acute where the seeding is non-uniformly distributed such as in regions of density variation and entrainment of unseeded ambient fluid [124], both of which occur in the current investigation.

Velocity biasing can be avoided by sampling at regular intervals when the interval is equal to at least twice the integral time scale to ensure statistically independent samples. Two such methods are the controlled processor [125] where time is divided into regular intervals and only the first sample in each period is recorded, or the similar ‘dead time’ sampling technique where signals are ignored for a fixed period after recording a signal [112]. If the individual samples are time stamped, similar results can be achieved post-facto by resampling the data gathered in burst mode at regular intervals. Such methods are only suitable where data rates are high. In the current investigation data rates near the wall, where velocities and seeding concentrations are low (when measuring in front of the nozzle lip), and methods that reject samples are not feasible due to rig operating time constraints, particularly when operating in ‘blow down’ mode.

In order to use all recorded samples a post-facto method is required to reduce the effects of the bias in the form of data weighting functions. Numerous such weighting methods have been suggested. Examples of weighting methods are: the inverse of the particle’s instantaneous velocity magnitude [122], residence or transit time weighting (the time taken by the particle to cross the measurement volume [126], particle interarrival time (time between consecutive particles being sampled) [126] and transformations of the skewed probability density function to a Gaussian distribution [127].

A review panel was established in order to evaluate biases and suggested correction techniques. The report by Edwards [120] concluded that the correction technique proposed by McLaughlin and Tiederman should not be used as the errors after correction could be greater than if no correction were applied at all. The use of inverse velocity magnitude weighting in the current experiments has been discounted as it is not suitable for the high speed and highly turbulent flows under investigation (as shown by the work of Gould and Loseke[128], Hoesel and Rodi[126] and Herrin and Dutton [129]). The use of particle interarrival time, as recommended by Edwards, and Herrin and Dutton is also not suitable due to the low data rates near to the wall.

Edwards recommended the use of transit time weighting for all data densities with the proviso that the seeding density was spatially uniform and the processing system was capable of accurately measuring the transit time. However, Herrin and Dutton encountered problems when using transit time weighting in their high speed tests due to the resolution of time
measurements. Their worst case transit time uncertainty was ±16%. With the LDA system used for the current measurements the F80 processor is able to resolve times to about 10ns. In the current investigation, for a velocity of 310m/s (which would occur at the nozzle exit when choked and operating at a total temperature of 288K), the residence time corresponds to 480ns without beam expanders and 239ns with beam expanders. This corresponds to a transit time uncertainty of only ±2% and ±4%. As data densities in the current investigation are typically very low in the near-wall region, transit time weighting was used in favour of particle interarrival time during post-facto reduction of velocity bias for processing of the LDA measurements.

2.5.2.3 Velocity Gradient Broadening

A significant error source when measuring highly accelerated boundary layers using the LDA technique is velocity gradient broadening. Velocity gradient broadening occurs when the finite LDA measurement volume is located in regions of the flow where there are large mean velocity gradients. Such an example is the near wall region of boundary layers. The existence of a velocity gradient across the measurement volume means that the particles crossing the measurement volume will have a range of velocities in addition to the variations that exist due to velocity fluctuations. This has the effect of broadening the probability density function and the distribution becomes skewed.

The influence of large velocity gradients on LDA measurements was investigated by Durst et al. [124] who presented measurements of mean velocity and turbulence statistics of a pipe flow with a pipe Reynolds number of 22000 with varying measurement volume sizes. Increasing the measurement volume size significantly increased the deviation of the measured velocity profiles, with the measured profiles becoming flatter with increasing measurement volume scale. The greatest variation occurred in the near wall region. The range of values increases as the range of particle velocities increase, with large over-prediction of turbulence levels close to the wall, and non-zero values at the wall itself.

Durst et al. [124] suggested corrections for the effects of velocity gradient broadening based on the probability of a particle being detected being a function of the Gaussian nature of the light intensity distribution across the measurement volume. Durst expressed the measured velocity as a truncated Taylor series expansion about the geometric centre of the measurement volume such that:

\[ \langle U \rangle = U(y_c) + \frac{d^2}{32} \left( \frac{dU}{dy^2} \right) + \ldots \]  \tag{2.51}

\[ \langle u^2 \rangle = u^2(y_c) + \frac{d^2}{16} \left( \frac{dU}{dy} \right)^2 + \ldots \]  \tag{2.52}

The analysis was extended by Compton [130] to include

$$\langle v^2 \rangle = v^2(y_c) + \frac{d^2}{16} \left( \frac{dV}{dy} \right)^2 + \ldots \quad (2.53)$$

and

$$\langle uv \rangle = uv + \frac{d^2}{32} \left( \frac{d^2uv}{dy^2} \right) + \ldots \quad (2.54)$$

where \( d \) is the diameter at the waist of the measurement volume, based on the Gaussian distribution of light intensity. The analyses of Durst and Compton assume that the significant velocity gradients are only across the waist of the measurement volume and that gradients along the major axis are negligible.

In the current investigation, velocity gradient broadening arises from two sources. The first is the large radial gradients that exist across the region of the axisymmetric boundary layer that it covered by the waist of the ellipsoidal measurement volume. The second source of velocity gradients occurs due to the circumferential curvature of the boundary layer due to the round nozzle cross-section. An illustration of the measurement volume superimposed on the nozzle exit boundary layer in Figure 2.25 shows that large gradients exist not only across the waist of the measurement volume but also along its major axis due to the linear nature of the measurement volume, thus making the above corrections difficult to apply. Hence, to assess the importance of this error on the measurements taken in this thesis, the following approach has been taken. The influence of the measurement volume on the measured profiles was minimised using the beam expanders (discussed in Section 2.3) to reduce both the waist and length of the measurement volume. Data taken with several measurement volume sizes to assess its influence are reported below in Chapter 4.

2.6 Closure

The design, operation and modifications to Loughborough University’s High Pressure Nozzle Test Facility used during the measurement of nozzle inlet and nozzle exit boundary layers in the current work has been discussed. The two independent measurement techniques used to catalogue such boundary layers have been detailed. The methodologies followed as part of a best practice approach to the measurement of the boundary layers were described in Sections 2.2 and 2.3.

Due to the difficulties of studying compressible and highly accelerated boundary layers where boundary layers may be very thin and subjected to large favourable static pressure
2.6 Closure

gradients, unconventional methods of measuring and processing data were required, such as the chosen measurement location of the nozzle exit boundary layers (just downstream of the nozzle exit) and the use of kinematic boundary layer parameters to simplify the analysis and comparison of data. Such methods have been described and justified in Section 2.4. The major sources of errors inherent in the measurement techniques were detailed in Section 2.5 in an attempt to quantify experimental errors and to provide the best means of error control and correction for the results to be presented in Chapter 4.
2.7 Figures
A Pneumatic G4 Valve
B Globe Valve
C Tertiary Flow Tank Isolator Valve
D Air Receivers
E Desicant Air Dryer
F Settling Tank
G Oil Separator
H Intercooler
I W type compressor
J Tertiary Air Supply
Figure 2.2: High Pressure Nozzle Test Facility - Nozzle Test Rig

Figure 2.3: Schematic of the Nozzle Test Rig
Table 2.3: Key to Figures 2.2 and 2.3

<table>
<thead>
<tr>
<th></th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>A</td>
<td>Primary Nozzle Delivery Duct</td>
</tr>
<tr>
<td>B</td>
<td>Secondary Flow Plenum and Contraction</td>
</tr>
<tr>
<td>C</td>
<td>Combustor</td>
</tr>
<tr>
<td>D</td>
<td>Primary Control Valve</td>
</tr>
<tr>
<td>E</td>
<td>Globe Valve</td>
</tr>
<tr>
<td>F</td>
<td>Secondary Flow Supply Pipe</td>
</tr>
<tr>
<td>G</td>
<td>Rig Air Supply Pipe</td>
</tr>
<tr>
<td>H</td>
<td>Secondary Control Valve</td>
</tr>
</tbody>
</table>

Figure 2.4: HPNTF Control Room
Figure 2.5: LU48 and LU60 Nozzle Geometries

Figure 2.6: LU48(P) and LU60(P) Nozzle Geometries
Figure 2.7: Boundary Layer Pitot Probe Sensing Head

Figure 2.8: Inlet Measurement Extension Duct
Figure 2.9: Nozzle Inlet Boundary Layer Traverse
2.7 Figures

Figure 2.10: Nozzle Inlet Total Pressure Variation (LU60, NPR 1.57)

Figure 2.11: Nozzle Exit Total Pressure Variation (LU60, NPR 1.58)
Figure 2.12: Dual Probe for Nozzle Exit Measurements

Figure 2.13: Nozzle Exit Probe Traverse
2.7 Figures

Figure 2.14: Variation of Mean Velocity Profiles with Axial Distance [104]

Figure 2.15: Variation of RMS Velocity Profiles with Axial Distance [104]
Figure 2.16: LDA Solid Particle Seeder Unit (left), Liquid Droplet Seeding Unit (right)

Figure 2.17: LDA Probe and Traverse
Figure 2.18: LDA Transmitter Probe Orientation

Figure 2.19: Interpolation of Pressure Profile to Determine Effective Wall Location (LU60, NPR ≈ 1.6)
Figure 2.20: Interpolation of Velocity Profile to Determine Effective Wall Location ($LU_{60}$, $NPR \approx 1.6$)
Figure 2.21: Determination of Skin Friction Using the Clauser Plot Method

Figure 2.22: Influence of Probe Incidence on Measured Pressure
Figure 2.23: Mean Velocity Profile With Error Bars for 99% Confidence Interval ($LU60$, $NPR ≈ 1.68$)

Figure 2.24: RMS Axial Velocity Profile With Error Bars for 99% Confidence Interval ($LU60$, $NPR ≈ 1.68$)
Figure 2.25: LDA Measurement Volume
Chapter 3

Numerical Modelling

Methodology

3.1 Introduction

The motivation for understanding the state and shape of the nozzle exit boundary layer has been described in Chapter 1. The literature review highlighted factors that influence the state of the nozzle exit boundary layer, such as the state of the boundary layer at entry to the nozzle and, in particular, the large favourable pressure gradient that exists within the nozzle itself. Although the boundary layer at inlet to and exit from the nozzle is to be measured directly (as described in Chapter 2), it was not possible to investigate the development of the boundary layer as it negotiated the nozzle for practical reasons. In order to explain any observed variation between nozzle inlet and exit measurements, a numerical approach was adopted.

The numerical predictions in this thesis used an existing CFD code, the DELTA CFD code, developed within the Department of Aeronautical and Automotive Engineering at Loughborough University [131]. DELTA adopts a finite volume approach for the solution of the governing flow equations using a pressure-correction method and is based on a multiblock structured, curvilinear grid with a collocated storage approach and Rhie-Chow smoothing [132] to avoid pressure-velocity decoupling. For complex geometries a multiblock approach is adopted, allowing numerous structured blocks to be joined in an unstructured manner to avoid 'blockage' cells; the multiblock method also provides the user with greater control of mesh quality. DELTA was primarily written to simulate compressible, turbulent flows using a high Reynolds number, two-equation $k - \epsilon$ turbulence model in conjunction with
3.2 Governing Equations

The flow of a compressible, Newtonian, calorically perfect gas is governed by three fundamental physical laws: Conservation of Mass (often referred to as continuity), Conservation of Momentum (Newton’s Second Law of Motion) and Conservation of Energy (1st law of
3.2 Governing Equations

thermodynamics). Collectively, when applied to a fluid continuum, these three equations are referred to as the Navier-Stokes equations.

The conservative forms of these equations for a compressible flow can be written in Cartesian tensor notation as:

Conservation of mass (continuity):

\[
\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x_j} (\rho u_j) = 0
\]  

(3.1)

Conservation of momentum:

\[
\frac{\partial}{\partial t} (\rho u_i) + \frac{\partial}{\partial x_j} (\rho u_i u_j) = -\frac{\partial p}{\partial x_j} + \frac{\partial}{\partial x_j} (\tau_{ij})
\]  

(3.2)

where the viscous stress tensor \(\tau_{ij}\) is given by:

\[
\tau_{ij} = 2\mu S_{ij} - 2\frac{\partial}{\partial x_k} \delta_{ij} \frac{\partial n_k}{\partial x_k}
\]  

(3.3)

where \(\delta_{ij}\) is the Kronecker Delta (\(\delta_{ij} = 1\) if \(i = j\) and \(\delta_{ij} = 0\) if \(i \neq j\)) and the strain rate tensor \(S_{ij}\) is given by:

\[
S_{ij} = \frac{1}{2} \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right)
\]  

(3.4)

Conservation of Energy (written in total enthalpy form: \(H = E + \frac{\rho}{\rho}E\), where \(E\) is the total energy \(E = e + \frac{1}{2}u_iu_i\) and \(e\) is the specific internal energy):

\[
\frac{\partial (\rho H)}{\partial t} + \frac{\partial}{\partial x_j} (\rho u_j H) = -\frac{\partial p}{\partial t} + \frac{\partial}{\partial x_j} (u_i \tau_{ij} + q_j)
\]  

(3.5)

where the heat flux \(q_j\) is:

\[
q_j = -\frac{C_p \mu}{\sigma} \frac{\partial T}{\partial x_j}
\]  

(3.6)

and \(\sigma\) is the non-dimensional Prandtl number, defined as:

\[
\sigma = \frac{\mu C_p}{k}
\]  

(3.7)

where \(k\) is the coefficient of thermal conduction. In order to close the Navier-Stokes equations, the fluid is assumed to be a perfect gas (which assumes that intermolecular forces are negligible [136]) and, therefore, follows the equation of state:

\[
p = \rho RT
\]  

(3.8)
Where $R$ is the gas constant. The fluid is also assumed to be calorically perfect, thus allowing the internal energy to be related to static temperature:

$$ e = C_v T $$

where the gas constant is related to the fluid specific heats by:

$$ R = C_p - C_v $$

$$ \gamma = \frac{C_p}{C_v} $$

allowing the temperature to be defined in terms of the fluid's velocity and energy.

### 3.2.1 Time-Averaging of the Navier-Stokes Equations

For the majority of highly turbulent, engineering flows it is not feasible to solve the Navier-Stokes equations directly (as is done in Direct Numerical Simulations) as this would require numerical resolution of all of the turbulent scales, which is computationally prohibitively expensive at high Reynolds numbers; turbulence modelling therefore is required. In the current work a time-averaged approach to modelling of the Navier-Stokes equations was employed, whereby the influence of the turbulence on the flow is characterised via statistical quantities, an approach first suggested by Reynolds in 1895 [137]. This statistical approach is the foundation of a broad class of turbulence models, which are often referred to as Reynolds Averaged Navier-Stokes (RANS) models. DELTA uses a mass-weighted, time-averaged approach for all flow variables with the exception of density and pressure which require conventional time-averaging. The use of mass-weighting, often referred to as Favre averaging, yields equations of simpler form compared to conventional time-averaging [138]. In variable density flows, such as the high speed boundary layers investigated in this thesis (where density clearly varies due to the effects of compressibility) the use of mass-weighting also avoids the appearance of extra correlations between density and velocity fluctuations. The use of mass-weighted averaging for compressible flows as adopted in the DELTA code was applied to the study of high NPR jet flows by Birkby [139].

For Favre averaging, the instantaneous value can be decomposed as:

$$ \phi (x_i, t) = \bar{\phi} (x_i) + \phi^\prime (x_i, t) $$

where

$$ \bar{\phi} = \frac{1}{\rho} \lim_{T \to \infty} \frac{1}{T} \int_{t_0}^{t_0+T} \rho(x_i, t) \phi (x_i, t) \, dt $$

$$ \phi^\prime = \phi (x_i, t) - \bar{\phi} (x_i) $$
3.3 Turbulence Modelling

(where the overbar denotes a time-averaged quantity). For further details on mass-weighted averaging, the reader is referred to [138] and [140].

The time-averaged form of the Navier-Stokes equations can be written as:

**Continuity:**

\[
\frac{\partial \bar{\rho}}{\partial t} + \frac{\partial}{\partial x_j} (\bar{\rho} \bar{u}_j) = 0
\]  

(3.14)

**Conservation of Momentum:**

\[
\frac{\partial \bar{p} \bar{u}_i}{\partial t} + \frac{\partial}{\partial x_j} (\bar{p} \bar{u}_i \bar{u}_j) = -\frac{\partial \bar{p}}{\partial x_i} - \frac{\partial}{\partial x_j} \left( \bar{p} \bar{u}_i \bar{u}_j \right) + \frac{\partial}{\partial x_j} \left( \bar{\tau}_{ij} \right)
\]  

(3.15)

**Conservation of Total Enthalpy:**

\[
\frac{\partial}{\partial t} \left( \bar{p} \bar{H} \right) + \frac{\partial}{\partial x_j} \left( \bar{p} \bar{u}_j \bar{H} \right) = \frac{\partial \bar{p}}{\partial t} + \frac{\partial}{\partial x_j} \left( \bar{u}_i \bar{\tau}_{ij} + \bar{u}_i \bar{\tau}_{ij} - \bar{u}_i - \bar{p} \bar{u}_j \bar{H} \right)
\]  

(3.16)

**3.2.2 Constant Enthalpy Assumption**

Several assumptions have been made in the current work that greatly simplify the implementation of the total enthalpy equation. It is assumed that for the nozzle flows of interest here that the total temperature (and, hence, also the total enthalpy) of the flow at entry to the nozzle is constant and equal to the total temperature of the ambient fluid entrained by the jet after exit from the nozzle (which is equal to ambient static temperature for a stagnant ambient). As a consequence, if it is only the steady flow which is of interest, and assuming that the last term in Equation 3.16 is small, this implies that the total enthalpy is constant following any streamline and, hence, is constant throughout the whole flow domain.

This allows the solution of the energy equation (assuming \( \gamma \) remains constant) to be written directly as:

\[
\bar{H} = \frac{\gamma \bar{\rho}}{\gamma - 1} \bar{p} + \frac{1}{2} \bar{u}_i \bar{u}_i + \frac{1}{2} \bar{u}_i \bar{u}_j = \text{constant}
\]  

(3.17)

which removes the requirement for the solution of a transport equation for enthalpy, reducing computational overhead.

**3.3 Turbulence Modelling**

The application of mass-weighted time-averaging to the Navier-Stokes equations results in the appearance of additional unknown correlation terms, the turbulent Reynolds stresses
In order to arrive at a closed set of equations, modelling of the Reynolds stresses and turbulent fluxes is required. In the current work turbulence closure is achieved using a low Reynolds number form of the $k - \varepsilon$ equation, which is discussed in the following section.

### 3.3.1 $k - \varepsilon$ Turbulence Model

The $k - \varepsilon$ model is an eddy viscosity model based on an analogy between the behaviour of viscous stresses and turbulent stresses. As shown above, the viscous stresses are proportional to the mean rate of strain of a fluid element. Experimentally, it is observed that turbulence decays in the absence of mean shear and turbulent stresses increase with increasing mean rates of strain, similar to the viscous stresses [141]. This leads to Boussinesq's postulation that the turbulent stresses may be related to the strain rate of a fluid element (the Eddy Viscosity Hypothesis [29]) such that:

$$
-\bar{p}\overline{u_i u_j} = \mu_t \left( \frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i} \right) - \frac{2}{3} \delta_{ij} \left( \mu_t \frac{\partial \bar{u}_k}{\partial x_k} + \bar{p} \right)
$$

(3.18)

Where $\mu_t$ is the eddy or turbulent viscosity.

In the $k - \varepsilon$ turbulence model, the turbulent viscosity can be written as:

$$
\mu_t = \bar{p} C_{\mu} \frac{k^2}{\varepsilon}
$$

(3.19)

which requires two additional transport equations to be solved for $\bar{k}$ and $\varepsilon$.

The high Reynolds number form of the modelled transport equations for equations are:

$$
\frac{\partial (\bar{p} \bar{k})}{\partial t} + \frac{\partial (\bar{p} \bar{u}_j \bar{k})}{\partial x_j} = \frac{\partial}{\partial x_j} \left( \left( \frac{\mu_t}{\sigma_k} \right) \frac{\partial \bar{k}}{\partial x_j} \right) + P_k - \bar{p} \varepsilon
$$

(3.20)

and

$$
\frac{\partial (\bar{p} \varepsilon)}{\partial t} + \frac{\partial (\bar{p} \bar{u}_j \varepsilon)}{\partial x_j} = \frac{\partial}{\partial x_j} \left( \left( \frac{\mu_t}{\sigma_{\varepsilon}} \right) \frac{\partial \varepsilon}{\partial x_j} \right) + \frac{\varepsilon}{k} \left( C_{\varepsilon 1} P_k - C_{\varepsilon 2} \bar{p} \varepsilon \right)
$$

(3.21)

where the production of turbulence kinetic energy $P_k$ is given by:

$$
P_k = \bar{p} \overline{u_i u_j} \left( \frac{\partial \bar{u}_i}{\partial x_j} \right)
$$

(3.22)

and $C_{\mu}, C_{\varepsilon 1}, C_{\varepsilon 2}, \sigma_k, \sigma_{\varepsilon}$ are empirical constants as defined by Launder and Spalding [142] and given in Table 3.1.
3.3 Turbulence Modelling

<table>
<thead>
<tr>
<th>$C_\mu$</th>
<th>$C_{l1}$</th>
<th>$C_{l2}$</th>
<th>$\sigma_k$</th>
<th>$\sigma_\epsilon$</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.09</td>
<td>1.44</td>
<td>1.92</td>
<td>1.0</td>
<td>1.3</td>
</tr>
</tbody>
</table>

Table 3.1: $k - \epsilon$ Turbulence Model Coefficients

In the current investigation, it is the development of the boundary layer within the nozzle which is of prime interest. For High Reynolds number flows, where viscous effects are considered unimportant ($\mu_t >> \mu$), the standard $k - \epsilon$ model (above) avoids the necessity of integrating across the whole of the boundary to the wall by applying wall functions to bridge the viscous dominated region of the boundary layer. The wall functions are based on the universal behaviour of the boundary layer near the wall (the logarithmic law of the wall and turbulent equilibrium, as discussed in Section 1.5) avoiding the need to resolve the near-wall region fully which requires very fine meshes and is, therefore, computationally expensive. However, in many flow situations the assumptions used in defining the logarithmic behaviour of the boundary layer are not valid. Relevant examples of such situations are low Reynolds number [67] and highly accelerated (relaminarising) flows which were shown to depart from the universal law of the wall in Section 1.6.

In such cases extension of the high Reynolds number form of the $k - \epsilon$ model is required to capture correctly the viscous-dominated near-wall behaviour. Any modifications must revert to the high Reynolds number form when the effects of viscosity become negligible. In this thesis, the low Reynolds number extension to the high Reynolds number $k - \epsilon$ model, proposed by Launder and Sharma [96] is adopted and is described below.

3.3.1.1 Launder-Sharma Low Reynolds Number Turbulence Model

The Launder-Sharma low Reynolds number model introduces additional terms for near-wall viscous diffusion, damping functions which are functions of the turbulence Reynolds number:

$$Re_t = \frac{k^2}{\mu_\epsilon} \propto \frac{\mu_t}{\mu}$$

(3.23)

and additional terms to improve the model’s ability to capture the near-wall behaviour. The following section briefly describes the Launder-Sharma model. For a more detailed review of low Reynolds number models and the behaviour of the individual functions the reader is directed to the works of Patel et al. [95], Leschziner and Drikakis [143] and Wilcox [144].

The Launder-Sharma form of the $k$ and $\epsilon$ transport equations become:

$$\frac{\partial (\rho k)}{\partial t} + \frac{\partial (\rho u_j k)}{\partial x_j} = \frac{\partial}{\partial x_j} \left( \mu + \frac{\mu_t}{\sigma_k} \frac{\partial k}{\partial x_j} \right) + P_k - \rho (\ddot{\epsilon} + D)$$

(3.24)
3.3 Turbulence Modelling

\[
\frac{\partial (\tilde{\rho} \tilde{\varepsilon})}{\partial t} + \frac{\partial}{\partial x_j} (\tilde{\rho} \tilde{u}_j \tilde{\varepsilon}) = \frac{\partial}{\partial x_j} \left( \left( \mu + \frac{\mu_t}{\sigma_{\varepsilon}} \right) \frac{\partial \tilde{\varepsilon}}{\partial x_j} \right) + \frac{\tilde{\varepsilon}}{\tilde{k}} (C_{11}f_1 P_k - \tilde{\rho} C_{12}f_2 \tilde{\varepsilon}) + E \tag{3.25}
\]

and the turbulent viscosity becomes:

\[
\mu_t = \tilde{\rho} C_{\mu} f_\mu \frac{\tilde{k}^2}{\tilde{\varepsilon}} \tag{3.26}
\]

The mass-weighted dissipation rate \( \tilde{\varepsilon} \) has been replaced with the isotropic dissipation rate \( \tilde{\varepsilon} \). This was first introduced by Jones and Launder [94] as a numerical simplification, allowing a value of \( \tilde{\varepsilon} \) equal to zero to be specified as a wall boundary condition. Its use introduces the term \( D \), which is equal to the wall dissipation rate and is required to balance the turbulent kinetic energy at the wall:

\[
\tilde{\varepsilon} \bigg|_{y=0} = D \bigg|_{y=0} \tag{3.27}
\]

but tends asymptotically to zero in the fully turbulent region where the isotropic dissipation rate must equal the total dissipation rate [95]:

\[
\tilde{\varepsilon} = \tilde{\varepsilon} \tag{3.28}
\]

The function \( D \) is defined as:

\[
D = 2\mu \left( \frac{\partial \left( \frac{1}{\tilde{k}} \right)}{\partial x_j} \right) \tag{3.29}
\]

The function \( E \) in the transport equation for \( \tilde{\varepsilon} \) is defined as:

\[
E = 2\mu_t \left( \frac{\partial^2 \tilde{u}_i}{\partial x_j \partial x_k} \right)^2 \tag{3.30}
\]

and acts to increase the dissipation rate in the buffer region to correct peak turbulence levels.

The damping function \( f_\mu \) is the most important as it features in all of the transport equations. It’s purpose is to simulate the influence of molecular viscosity on the Reynolds shear stresses near the wall and is defined as:

\[
f_\mu = \exp \left[ -\frac{3.4}{\left( 1 + \frac{Re_{\varepsilon} \nu}{Re_{\varepsilon} \nu_0} \right)^2} \right] \tag{3.31}
\]
3.4 Generic Transport Equation

The damping function \( f_2 \) is used to include low Reynolds number effects in the destruction term of the \( \dot{\varepsilon} \) transport equation. Its effects are limited to the viscous sublayer, asymptoting to unity at \( Re_l = 15 \).

\[
f_2 = 1 - 0.3 \exp \left( -Re_l^2 \right)
\]  
(3.32)

The production term in the \( \dot{\varepsilon} \) transport equation remains unchanged from its high Reynolds number counterpart so its damping function is equal to unity:

\[
f_1 = 1
\]  
(3.33)

The model coefficients \( C_{u1}, C_{v1} \) and \( C_\omega \) remain unchanged from the high Reynolds number form and are given in Table 3.1.

3.4 Generic Transport Equation

The time-averaging of the Navier-Stokes equations results in six modelled transport equations: mass, momentum (three components), turbulent kinetic energy and turbulent dissipation. DELTA solves each of the transport equations in the same form, using a generalised transport equation, which greatly simplifies the numerical procedure. Each of the transport equations can be written in the general form:

\[
\frac{\partial \left( \tilde{\rho} \phi \right)}{\partial t} + \frac{\partial \left( \tilde{\rho} \tilde{u}_j \phi \right)}{\partial x_j} = \frac{\partial}{\partial x_j} \left( \Gamma \frac{\partial \phi}{\partial x_j} \right) + S_\phi
\]  
(3.34)

The source term and diffusion coefficient for each of the transport equations are given in Table 3.2.

3.5 Grid Generation and Geometry Definition

In order to solve the discretised forms of the governing equations, a body-fitted mesh is required. Throughout this thesis, generation of the structured, multiblock grids required by DELTA was performed using ICEM CFD Hexa version 5 [145]. Mesh generation starts with a CAD representation (see below) of the selected solution domain. The grid generator automatically generates an initial block around the whole geometry which can then be subdivided into smaller blocks and adjusted to fit the underlying geometry by the user to craft the desired shape. Hexa adopts a top-down approach whereby each time a block is
3.5 Grid Generation and Geometry Definition

<table>
<thead>
<tr>
<th>Transport Equation</th>
<th>Scalar $\phi$</th>
<th>Diffusion Coefficient $\Gamma_\phi$</th>
<th>Source $S_\phi$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Continuity</td>
<td>1</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>Momentum</td>
<td>$\bar{u}_i$</td>
<td>$\mu + \mu_t$</td>
<td>$-\frac{\partial \phi}{\partial x_i}$</td>
</tr>
<tr>
<td>Turbulent Kinetic Energy</td>
<td>$\bar{k}$</td>
<td>$\mu + \frac{\mu_k}{\sigma_k}$</td>
<td>$\mu_t \left( \frac{\partial \bar{k}}{\partial x_j} + \frac{\partial \bar{k}}{\partial x_i} \right) \frac{\partial \bar{k}}{\partial x_j} - \bar{p} (\bar{e} + D)$</td>
</tr>
<tr>
<td>Turbulent Dissipation</td>
<td>$\bar{\varepsilon}$</td>
<td>$\mu + \frac{\mu_\varepsilon}{\sigma_\varepsilon}$</td>
<td>$C_{t1} f_1 P_k \bar{k} \frac{\varepsilon}{k} - \bar{p} C_{t2} f_2 \frac{\varepsilon^2}{k} + E$</td>
</tr>
</tbody>
</table>

Table 3.2: Transport Equation Diffusion Coefficients and Source Terms

modified all other blocks connected to it are also modified [145]. Once a suitable blocking strategy has been established, the user can begin defining the mesh itself through a process of association (assigning vertices, edges and faces of the blocks with the relevant area of the physical geometry, in effect attaching the computed geometry to the physical geometry) and edge definition (where the number of nodes and their distribution along each block edge is defined). Structured mesh generation is often an iterative procedure, with numerous revisions of the grid being required to ensure adequate resolution, particularly in regions of high gradients within the flow, and mesh quality. Further details on grid generation can be found in [145].

3.5.1 Geometry Definition

A 3-dimensional representation of the domain geometry was required for the purpose of mesh generation. SolidEdge Version 15 [146] was used to create a solid model of the region that would contain the mesh. The use of a parametric modeller greatly simplified producing the geometric definitions, allowing modification to the design (such as changing nozzle diameters, delivery pipe lengths and adding transition trips) without the requirement of redrawing the whole body. Due to the axisymmetry of all of the geometries considered in this thesis the solution domains were defined as a 2-dimensional surface then extruded in the azimuthal direction to produce a 3-dimensional body of revolution. An example of a typical solid model of a geometry is shown in Figure 3.1. The figure shows a solid body representation of a 15° sector of a round delivery pipe feeding an axial conical nozzle with a short parallel extension added to its exit. The solid model represents the region in which the structured mesh will be generated. ICEMCFD includes numerous translators to convert foreign file formats into one suitable for the grid generator itself. The most satisfactory file format
found for the simple axisymmetric geometries used was the .stl or stereolithography format which is used as the standard input for most rapid prototyping machines. Once imported, the grid generator performs a user-driven decomposition of the surface of the solid model to define the points, surfaces and faces required during the process of association, described above.

3.6 Boundary Conditions

Several boundary condition types were used in this thesis, depending on the nature of the flow. As most boundary condition types and their implementation are well documented in standard texts on numerical methods for fluid flows, the following section provides only a brief description of the boundary conditions used, with particular focus on the modification of existing boundary conditions within DELTA required for the present work. For greater detail on the characteristic behaviour of the equations of fluid flows and boundary conditions the reader is directed to the work of Versteeg and Malalasekera [141], Ferziger and Peric' [147] and Anderson [136].

Boundary conditions used include:

- Fixed velocity (incompressible) and fixed total pressure (compressible) conditions at nozzle inlet planes. This involves fixing a specific value of either all velocity components or total pressure (and also turbulent kinetic energy and dissipation rate) at each grid node in the inlet plane. As the state and shape of the boundary layer at inlet to the nozzle is important in the current investigation, the inlet condition treatments within the DELTA code were modified to allow the user to apply specific profiles at the inlet. This allowed experimental data to be used (such as those measured at the nozzle inlet, as described in Chapter 2) or data from precursor calculations which removed the requirement to simulate large lengths of the delivery duct for each flow scenario in order to provide realistic conditions at the nozzle inlet. Where more detailed turbulence data were not available, uniform inlet values of $k$ and $\tilde{\varepsilon}$ were applied, based on assumed turbulent velocity and length scales estimated from a relevant length (typically the diameter of the inlet to the nozzle or duct) and turbulence intensity. Inlet profiles were interpolated based on wall normal distance using a cubic spline method [148] to determine values at the cell centres.

- Zero gradient and fixed static pressure conditions for nozzle exit and jet plume exit conditions. The zero gradient condition specifies all gradients in the exit boundary
normal direction to be equal to zero such that:

\[
\frac{\partial \phi}{\partial n} = 0
\]  

(3.35)

Where \( n \) denotes the normal direction. As the focus of the simulations is the development of the flow within the nozzle and not the development of the jet plume, most simulations were restricted to the nozzle only to avoid the additional computational overhead associated with computing the jet plume as well. For simulations of the flow within nozzles operating at pressure ratios which correspond to under-expanded conditions it was necessary to specify a nozzle exit static pressure, which was derived from gas dynamic relations and assumes an exit Mach number equal to unity (choked exit conditions). For these simulations a static pressure boundary condition was used to fix the exit static pressure. It is common practice when using fixed static pressure boundary conditions to locate these as far as practically possible away from the region of interest to obtain 'transparent' boundary conditions that do not influence the region of interest [141]. Unfortunately, in order to restrict the solution to the nozzle only, the static pressure boundary was located at the nozzle exit plane, close to the region of interest. The impact of the location of the domain exit boundary for nozzle simulations is addressed in Chapter 5 by a simulation which included both nozzle flow and jet plume flow. For this simulation entrainment pressure conditions were applied on the inlet, exit and far field boundaries of the solution domain.

- Mirror symmetry conditions. The use of symmetry conditions allows the user to take advantage of the axisymmetry of all of the simulated geometries which allows significant savings in computational effort to be made through removing the need to simulate the whole of the geometry. The symmetry boundary condition ensures no flow or scalar flux across the boundary and can be described by:

\[
u_n = 0
\]  

(3.36)

and:

\[
\frac{\partial \phi}{\partial n} = 0
\]  

(3.37)

This was used for the boundary conditions on the sector sides of the geometry shown in Figure 3.1.

### 3.7 Solution Approach

For all of the simulations second order accurate discretization of the convective fluxes in the momentum equation and the \( k \) and \( \tilde{c} \) equation was achieved using a limited form of
the Quadratic Upwind Interpolation for Convective Kinetics (QUICK) differencing scheme [131]. As a time-accurate solution was not required a spatially varying timestep based on the Courant-Friedrichs-Lewy (CFL) constraint was adopted to avoid numerical stiffness. In the current work, typical CFL numbers ranged from 1 to 3.

The solution was deemed to have converged once the residual error had decreased by several orders of magnitude and leveled out. As the rate of decrease of the residuals was often slow (with convergence taking on the order of 100000 iterations) convergence was also judged by comparison of profiles of mean velocity and turbulence statistics taken after varying number of iterations.

3.8 Code Validation

As stated in Section 3.1, DELTA has previously been demonstrated to capture highly turbulent and compressible flows with reasonable levels of accuracy. The focus of this section is, therefore, the verification of the implementation of the low Reynolds number extension and assessment of this model's ability to capture low Reynolds number boundary layers and effects such as relaminarisation. Two test cases were selected which typify relevant low Reynolds number phenomena. The first test case is the fully developed, turbulent pipe flow experiment at a low Reynolds number of Kudva and Sesonske [135], one of the simplest flows which will allow the model to be tested independently of large pressure gradients and complex geometries. A previous study of the Launder-Sharma model by Bardina et al. [149] highlighted the sensitivity of the prediction to mesh resolution and the distance between the wall and the first grid node. The simplicity of this test case is also well suited to assessing grid sensitivity in the wall normal direction. The second test case was designed to assess the capability of the code to predict the highly accelerated axisymmetric boundary layer presented in the detailed study of Fernholz and Warnack [74], which is representative of the conditions that may be encountered in contracting propulsion nozzles.

3.8.1 Low Reynolds Number Pipe Flow

The Low Reynolds number isothermal measurements of Kudva and Sesonske [135] were conducted using a vertical pipe arrangement with an internal diameter of 0.0364m and a length of 4.88m, which corresponded to a length-to-diameter ratio of 132. Ethylene Glycol was used as the working fluid and the tests were conducted at a pipe Reynolds number of 6032 based on bulk-averaged velocity and viscosity. Measurements of the mean and fluctuating
components of the stream-wise velocity were conducted using the hot film technique. The friction velocity \( u_\tau \) was estimated from the Blasius equation for the friction factor.

### 3.8.1.1 Geometry, Mesh and Boundary Conditions

The size of the pipe, bulk velocity and fluid viscosity were chosen to allow an identical pipe Reynolds number as used in the experimental work. A pipe with a diameter of 0.005m and 1m in length was selected. A 15° sector was chosen as the cross-section of the domain. In order to alleviate the convergence problems associated with a polar structured mesh converging on a singularity at the centreline, a small ‘root’ boundary with a slip flow condition, with a radius equal to 1% of the pipe radius was included to ensure a finite surface area of the cell faces at the centreline. Several meshes were used, primarily to investigate the influence of the number of radial mesh points on the predicted profiles. Each mesh featured 100 cells uniformly distributed in the axial direction, and 10 uniformly distributed in the azimuthal direction. Although all of the flows under investigation in this thesis demonstrate axisymmetry and the resolution of the mesh in the azimuthal direction should not be important, the work of Hughes [150] has shown that the mesh resolution in the azimuthal direction is significant solving for Cartesian decomposed velocity components in what is essentially a cylindrical-polar problem. This stems from use of straight lines to represent curved surfaces, reducing the effective surface area over which the wall shear stress acts. 10 azimuthal cells within a 15° section were sufficient to avoid this problem. Four different radial resolutions were used. Three meshes had an identical non-dimensional distance between the wall and the first cell centre of 0.1\( y^+ \), based on the recommendations of Bardina et al. [149] that the location of the first cell should be limited to less than 0.3\( y^+ \). The total numbers of grid points in the radial direction for these three meshes were 25, 50 and 100 with varying expansion ratios between 1.05 and 1.10. A fourth mesh was tested with 50 points uniform distributed in the radial direction which corresponded to a value of 1.1\( y^+ \) for the first cell centre to examine the influence of the wall normal distance of the first cell centre. The predicted solution at the pipe exit was recycled as inlet conditions until a fully developed solution was obtained.

### 3.8.1.2 Results and Discussion

The semi-logarithmic plot of the mean axial velocity in Figure 3.2 shows that all of the meshes resolve the behaviour in the viscous sub-layer and predict the expected overshoot of the profile above the standard log-law line in the logarithmic region, a fundamental feature of low Reynolds number turbulent boundary layers. Beyond the edge of the viscous region there are noticeable differences between results from meshes with 50 and 100 radial points and the meshes with 25 points and 100 points but uniformly distributed. The cases with
50 and 100 points follow the experimental profile very closely through the buffer region. These profiles contained 25 and 43 points within the $y^+ < 30$ region for the 50 and 100 point meshes. A slight overshoot of the experimental data is observed in the logarithmic region with the gradient in the log-law region being greater than measured experimentally. Negligible differences exist between the predictions of the mean flow on both meshes, suggesting that predictions are grid independent. The same cannot be said of the mesh with 25 points which can be seen to depart from the experimental data beyond the edge of the viscous sub-layer ($y^+ \approx 10$) despite the same first cell spacing. Beyond the end of the transition region a clearly logarithmic region with a similar gradient to the 50 and 100 point cases can be seen, but it is displaced below the experimental curve. A similar behaviour is shown by the results of the mesh with 100 uniformly distributed points, again departing from the experimental data at the edge of the viscous sub-layer. Both of these meshes had a similar number of points within the $y^+ < 30$ region, (12 and 14 points for the 25 and 100 (uniform) meshes, respectively). Also included in Figure 3.2 are the results of the predictions of the same case using the Launder-Sharma model as presented by Craft et al. [151]. The results of the cases with 50 and 100 points are in good agreement with those of Craft et al. whose results also show a similar overshoot of the experimental data at the outer edge of the boundary layer. This agreement provides extra confidence in the current implementation of the Launder-Sharma model.

Figure 3.3 shows the predicted Reynolds stress profiles ($\rho u'v'$), non-dimensionalised with respect to the wall friction velocity $u_\tau$. The results show all of the different meshes capturing the general behaviour, increasing from zero at the edge of the viscous sub-layer, reaching a peak of approximately 0.75 at $y^+ \approx 30$, which corresponds to the edge of the buffer region then decreasing in a linear manner to zero at the centreline of the pipe. The profiles corresponding to 50 and 100 radial points follow the experimental data very closely across the whole of the boundary layer, whereas the results from 25 and 100 (uniform) points overshoot the experimental data in the outer region and the buffer region. Predicted peak values are seen to decrease with increasing mesh resolution but variations between the 50 and 100 point profiles are small, approximately 2%.

These results show that the Launder-Sharma model is capable of capturing the characteristics of a low Reynolds number boundary layer. The results show close agreement between numerical and experimental results for both mean and turbulent properties. The results also highlight the sensitivity of the model to near-wall grid resolution. The results agree with the findings of Bardina et al. [149] that the location of the first cell centre and number of points within the boundary layer is important. The results from the meshes with 50 and 100 radial points with a $y^+ << 1$ support their recommendations of $y^+ < 0.3$ for the first grid node and the use of approximately 60 mesh points across the boundary layer. The results
also suggest the region where mesh resolution is most significant is the buffer region, beyond which the flow becomes momentum dominated and the turbulence model should revert to the high Reynolds number form.

### 3.8.2 Relaminarising Axisymmetric Boundary Layer

The ability of the Launder-Sharma model to simulate relaminarisation was assessed against the experimental study of highly accelerated boundary layers of Fernholz and Warnack [89] and Warnack [89][152]. Tests were conducted in an axisymmetric low speed, low turbulence wind tunnel of 0.441m diameter (Figure 3.4). Accelerations of different strength were generated using movable, axisymmetric centre-bodies, located by radial rods attached at the rear of the centre-bodies so the boundary layer in the region of interest was not disturbed. Various initial boundary layer thicknesses were studied (defined at a datum axial location sufficiently far upstream from the centre-body to be unaffected by it). Boundary layer momentum thickness Reynolds numbers of different values were achieved by varying the bulk average inlet velocity (duct mass flow) and the length of the delivery pipe in which the boundary layer developed before entering the centre-body region. Boundary layers were assumed to be fully turbulent with momentum thickness Reynolds numbers in the range $Re_\theta 862 - 2564$. The boundary layers were tripped using velcro strips mounted 0.24m downstream of the leading edge of the test section in order to control transition. Measurements were conducted using hot wires and several method of measuring skin friction (Preston tubes, wall hot wire, oil film interferometry and a surface fence) to overcome the criticism of skin friction measurements made by Narashima and Sreenivasan [80][72]. In order to maintain equivalent test conditions on different days Warnack varied the inlet velocity to maintain a constant unit Reynolds number, defined as:

$$Re_{\text{unit}} = \frac{U_{\text{duct}}}{\nu}$$  \hspace{1cm} (3.38)

The velocity and viscosity were measured at datum points located upstream of the centre-body, away from the influence of the centrebody. Duct velocity ($U_{\text{duct}}$) was derived from Pitot probe measurements in the core region at the datum location and static pressure data from a wall static tapping located at 0.246m upstream of the inlet to the duct where the boundary layers were known to be thin. Details of boundary layer parameters at the datum locations have been provided [152] so these datum locations were adopted as the inlet locations for the main computations. For the present investigation case 2 was simulated. Experimental details of case 2 are presented in Table 3.3.
3.8 Code Validation

<table>
<thead>
<tr>
<th>$K_{\text{max}}$</th>
<th>$Re_{\text{inlet}}$</th>
<th>$H_{\text{inlet}}$</th>
<th>$U_{\text{inlet}}(\text{m/s})$</th>
<th>$\nu(\text{m}^2/\text{s})$</th>
<th>datum (m)</th>
</tr>
</thead>
<tbody>
<tr>
<td>$4.00 \times 10^{-6}$</td>
<td>862</td>
<td>1.48</td>
<td>7.45</td>
<td>$1.5505 \times 10^{-5}$</td>
<td>0.853</td>
</tr>
</tbody>
</table>

Table 3.3: Case 2 Experimental Conditions

3.8.2.1 Geometry, Mesh and Boundary Conditions

Once again, due to the axisymmetric nature of the experimental facility, it was not necessary to simulate the whole cross-section of the geometry; a $15^\circ$ section was again used, with 10 cells in the azimuthal direction. The solution domain extended axially from the datum location to beyond the end of the conical diffuser used in the experimental facility (Figure 3.4) to ensure that the zero gradient boundary condition at the domain exit was sufficiently far enough away from the region of interest as not to influence the flow. In order to use a polar mesh, a root of $2\text{mm}$ radius (approximately 1\% of the pipe radius), using a symmetry boundary condition was included.

The literature review in Chapter 1 noted that relaminarisation may be influenced by initial conditions, thus considerable effort was directed at ensuring that in the numerical simulations, the boundary layer conditions just upstream of the onset of acceleration agreed closely with experimental data. In order to reduce computational overheads, simulations were restricted to the region downstream of the datum locations used in the experimental investigation where free stream conditions and boundary layer parameters (velocity profiles, momentum thickness Reynolds number and turbulence profiles) were known [152] (see also Table 3.3). In order to produce accurate turbulence and velocity profiles for input at the inlet boundary of the solution domain, precursor calculations were conducted. This involved simulating a long length of the delivery pipe using a uniform velocity ($U_{\text{duct}} = 7.25 \text{m/s}$) boundary condition at its inlet and initial values of the turbulent kinetic energy and dissipation rate based on the pipe diameter and 0.2\% turbulence intensity (as measured in the tunnel without a centrebody by Warnack). In order to establish to what extent this precursor solution was independent of the grid, grid refinement tests were conducted by refining the mesh resolution in both the axial and radial directions. After grid-refinement the inlet profiles were simulated using a $100 \times 150 \times 10$ (axial, radial, azimuthal) mesh. Decisions on which axial station in the precursor calculation best matched the experimental data were made on the basis of comparing momentum thickness Reynolds number, free-stream velocity, shape factor and boundary layer thickness. The profiles finally selected using this approach for use as inlet conditions in the main simulations are shown in Figures 3.5-3.8. The axial velocity profile, presented in either boundary layer or log-law formats, is in good agreement with measured data. For turbulence properties the shear stress is well predicted, but the near-wall peak in $k$ is underpredicted, but the overall agreement for all properties was best.
3.8 Code Validation

\[
\begin{array}{|c|c|c|c|c|c|}
\hline
\text{Parameter} & \delta (\text{mm}) & U_0 (\text{m/s}) & \delta^* (\text{mm}) & \theta (\text{mm}) & H_{12} & R_{e_0} \\
\hline
\text{Experimental} & 16.51 & 7.44 & 2.649 & 1.796 & 1.476 & 862 \\
\text{LS Model} & 15.38 & 7.113 & 2.845 & 1.884 & 1.51 & 865 \\
\hline
\end{array}
\]

Table 3.4: Comparison of computational and experimental inlet conditions

The inlet profiles derived from the precursor calculations were then used as the inlet conditions for the main simulation using the user-defined profile inlet boundary condition for both mean and turbulent quantities.

The mesh used for the main simulation comprised of 200 \times 150 \times 10 cells in the axial, radial and azimuthal directions. As illustrated in Figure 3.9, high levels of skewness were encountered in the region close to the nose of the centrebody which was unavoidable due to the rapid convergence of the passage in this area. However, this was essentially an inviscid region of flow and the mesh shown was considered good enough. Care was taken to ensure mesh orthogonality in the boundary layer region on the outer pipe wall and the regions of high skewness were kept well away from the region of interest by creating a series of separate orthogonal blocks in the boundary layer region (Figure 3.10). The region of the mesh on the centrebody was deliberately kept coarse to reduce computational overhead as the boundary layer forming on its surface was expected to be thin (and therefore making minimal contribution to the blockage compared to the centrebody) and was not of interest.

Comparison between experimental and computational inlet boundary layer parameters are shown in Table 3.4. Note in particular that the momentum thickness Reynolds number and shape factor were well predicted and correspond to a turbulent boundary layer, although Figure 3.6 shows that the boundary layer is transitional as the log-law region is not fully developed.

3.8.2.2 Results and Discussion

Boundary layer integral parameters such as displacement and momentum thickness and associated Reynolds numbers were calculated using the methods described in the data reduction section in Chapter 2. The location of the edge of the boundary layer was not defined based on free stream velocity, instead it was defined using the same method used by Warnack [89] based on total and wall-static pressures. Warnack defined a non-dimensional dynamic pressure as:

\[
n_q = \frac{(P - p_{ref})}{(P_{ref} - p_{ref})}
\]

(3.39)
where $P$ is the local total pressure, and $P_{ref}$ and $p_{ref}$ are the core flow total and wall static pressures at an axial location of 0.246m (downstream of the location of the transition trip) taken from the experimental measurements. The edge of the boundary layer was then defined as the location where $n_q$ reached 0.995.

Figure 3.11 shows the variation of the acceleration parameter $K$, local skin friction coefficient, boundary layer shape factor ($H_{12}$) and momentum thickness Reynolds number through the contraction and relaxation regions of the flow. Included in the figures are the boundary layer parameters predicted using the standard $k-\varepsilon$ turbulence model with wall functions to demonstrate the need for the use of a low Reynolds number model to capture relaminarisation. Figure 3.11(a) shows that the variation of the acceleration parameter predicted by the standard $k-\varepsilon$ and Launder-Sharma models closely follows the experimental values. The small over prediction of the peak value by the Launder-Sharma model is attributed to small variations in the free stream velocity between experimental and computation conditions.

Figure 3.11(b) shows the variation of the local skin friction. As mentioned in the introduction, the skin friction is an important factor in identifying the onset of relaminarisation as a decrease denotes reversion from a turbulent to laminar-like state. The standard $k-\varepsilon$ model fails to capture the correct $c_f$ at onset of the acceleration and the general trend through the acceleration and relaxation regions, in particular, the rapid decrease in $c_f$ during relaminarisation, an important feature when identifying onset. The results of the Launder-Sharma model show an under-prediction of the skin friction in the initial stages of the acceleration. The same under-prediction was seen in the inlet profiles, as characterised in the overshoot of the predicted velocity profile beyond the experimental profile in the logarithmic region (Figure 3.6). However, the computed $c_f$ results closely follow the experimental trend, increasing through the contraction as the boundary layer is accelerated, reaching a maximum of $4.89 \times 10^{-3}$ at approximately 1.45m, followed by a very rapid decrease as reversion to the laminar-like state begins, reaching a minimum of $2.92 \times 10^{-3}$ at approximately 1.6m, the same location as measured experimentally. Beyond the minimum, the skin friction increases rapidly, once the acceleration has been relaxed and the boundary layer begins to revert back to the fully turbulent state. Although predictions follow the same trend as the experimental data, values of the skin friction are over-predicted and the second peak occurs sooner than in the experimental data, indicating a more rapid return to the turbulent state.

The variation of the shape factor $H_{12}$ is another important factor in assessing boundary layer relaminarisation, as it is an indicator of the boundary layer's state (a developed turbulent boundary layer has a shape factor of $H_{12} \approx 1.3$, whereas a much higher value, 2.6 for a Blasius profile, indicates a laminar profile). The predicted results are shown in Figure 3.11(c). The results of the standard $k-\varepsilon$ model do predict a small increase in $H_{12}$ at the onset of
the acceleration but the model then fails to capture the increase towards a more laminar-like value during relaminarisation and a decrease back towards a turbulent value once the acceleration is relaxed. The results of the Launder-Sharma model follow the experimental trend, decreasing in the initial stages as the boundary layer begins to respond to the acceleration, reaching a minimum of 1.41 at 1.35m, before increasing towards a more laminar-like value, peaking at 1.91 at approximately 1.62m (compared to the experimental peak value of 1.68 at 1.70m). It is noteworthy that in both experiment and numerical predictions, the shape factor is seen to begin to rise before the skin friction has reached a maximum suggesting that the increase in the shape factor does not necessarily mean that relaminarisation will occur. The peak in the predicted value is considerably larger and occurs at an earlier location than seen in the experimental data. Once the acceleration has been relaxed, the results follow the experimental trend, returning to a more turbulent value as the boundary layer returns to being fully turbulent. The predicted values show a more gradual change beyond 1.7m. The overprediction of the shape factor and underprediction of the local skin friction coefficient once the acceleration was relaxed is mirrored by the study of Viala and Aupoix [99] who saw similar behaviour when simulating the relaminarising flow of Blackwelder and Kovasznay [84]. Variation of the boundary layer momentum thickness Reynolds number is given in Figure 3.11(d). The results of the standard $k - \varepsilon$ model do follow the trend of the experimental data, decreasing rapidly during the acceleration and increasing rapidly once the acceleration is relaxed. Values are however consistently higher than the experimental data. The results of the Launder-Sharma model follow the experimental results very closely, decreasing as the boundary layer is accelerated and becoming thinner (reaching a minimum value of 284 at 1.60m compared to the experimental variation with a minimum value of 357 at 1.60m) but showing a slightly less rapid increase once the acceleration has ceased. Comparison of key values of $K$, $c_f$, $H_{12}$ and $Re_\theta$ from the experiments and Launder-Sharma model predictions are summarised in Table 3.5, with encouraging agreement.

<table>
<thead>
<tr>
<th></th>
<th>$K_{max}$</th>
<th>$Re_\theta$</th>
<th>$H_{12S}$</th>
<th>$Re_\theta_{MIN}$</th>
<th>$H_{12MIN}$</th>
<th>$H_{12MAX}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Experimental</td>
<td>$4.0 \times 10^{-6}$</td>
<td>862</td>
<td>1.48</td>
<td>357</td>
<td>1.34</td>
<td>1.68</td>
</tr>
<tr>
<td>LS model</td>
<td>$4.21 \times 10^{-6}$</td>
<td>865</td>
<td>1.51</td>
<td>284</td>
<td>1.41</td>
<td>1.90</td>
</tr>
</tbody>
</table>

Table 3.5: Comparison of Key Boundary Layer Parameters (Warnack, Case 2)

The above results have clearly demonstrated that, as expected, the standard $k - \varepsilon$ model is not capable of capturing relaminarisation. All subsequent discussion will therefore be restricted to predictions using the Launder-Sharma model.

Turning attention to the development of the boundary layer profiles, semi-logarithmic plots of radial profiles of mean axial velocity development are shown in Figure 3.12. The
profiles show the Launder-Sharma model following precisely the trends seen in the experimental data. At an axial location of 1.303m, the predicted profile shows the thickening of the viscous sublayer, but overshoots the experimental profile indicating an underprediction of the skin friction coefficient, as noted already (Figure 3.11(b)). As the severity of the acceleration increases, the extent of the overshoot increases due to the turbulence becoming less active and its interaction with the mean velocity profile becoming less significant. By 1.453m, the difference between the measured and predicted profiles has become much smaller, with the predicted profile closely following the experimental profile to the end of the buffer region which, due the thickening of the viscous sublayer, now corresponds to $y^+ \approx 50$. The profiles at 1.603m and 1.653m correspond to the relaminarising region of the flow and are characterised by a large overshoot of the standard log-law line and the complete absence of a logarithmic region (following a Blassius profile [74]). The predicted profiles show excellent agreement in this region.

The remaining profiles correspond to the relaxation of the acceleration and subsequent recovery of the boundary layer. In this region the Launder-Sharma model begins to depart from the experimental data. Most noticeable is the under-shoot of the experimental data (indicative of the over-prediction of local skin friction, as seen in Figure 3.11(b), and the premature reappearance of a logarithmic region and wake region. By 1.853m, approximately 0.15m beyond the end of the acceleration region, the experimental profile is seen to lie on the standard log-law line. The predicted results lie above the experimental profile in a similar manner to the profiles at inlet to the accelerated region (Figure 3.5).

Another important behaviour in a relaminarising boundary layer is the reduction in the peak Reynolds shear stress component $\overline{u'v'}$ (non-dimensionalised with respect to $u_2$) which becomes significant as the turbulence fails to respond rapidly to the acceleration, as described in Section 1.6.1. Variation of the radial profiles of Reynolds shear stress are shown in Figure 3.13. Unfortunately, despite the use of miniature crosswire probes, detailed measurements in the near-wall region below $y^+ \approx 60$ were not possible [91], but the general trends can still be assessed.

Although the predicted profiles follow the same general trend as the experimental profiles, (decreasing and moving toward the wall as the flow is accelerated, followed by a rapid increase and outward movement of the peak as the acceleration is relaxed) there is a very pronounced overprediction of the decay of the Reynolds stress in the accelerated region, which results in the formation of a double peak in the predicted profile. The double peak is most pronounced in the profile at 1.603m and 1.653m (where the boundary layer can be said to have relaminarised) where the Reynolds shear stress has decreased to almost zero for $30 < y^+ \leq 200$. Comparison of the mean profiles in Figure 3.12 and the Reynolds shear
stress profiles reveals that this region corresponds to the buffer region where the influence of the E term in the Launder-Sharma model is at its greatest [81]. Patel et al. noted that the turbulent kinetic energy in the buffer region was under predicted, which was attributed to the E term. This underprediction of $k$ will feed into the Reynolds shear stress through its dependence on the turbulent viscosity $\mu_t$.

The results of this relaminarisation test case have shown that the current implementation of the Launder-Sharma model is capable of satisfactorily predicting the phenomenon of boundary layer relaminarisation. The model closely predicted the variation of global boundary layer parameters and mean profile development in the acceleration. Predicted variations of the Reynolds stress $\overline{\rho u'v'}$ followed the correct trends, but showed over-damping within the buffer region. The model's ability to predict the development of the boundary layer once the acceleration has been relaxed is less satisfactory, overpredicting the rate of recovery of the boundary layer, but the model does capture the general trend of the events and this validation test case has produced sufficient confidence in the Launder-Sharma model to justify its use in the nozzle flow experimental geometries.

3.9 Closure

The numerical methods to be used to predict the boundary layer development through the nozzle have been presented in this chapter. The flow solver, DELTA has been described, as have the governing equations of fluid flow and the turbulence modelling required to achieve closure. Particular attention was paid to the addition of the Launder-Sharma low Reynolds number turbulence model to the DELTA code to allow capture of low Reynolds number effects.

Two validation tests cases were used to verify the implementation of the turbulence model and its ability to predict low Reynolds number boundary layers and relaminarisation. Comparison with experimental data showed that the model was capable of satisfactorily capturing boundary layer relaminarisation, both in terms of global boundary layer parameters and development of the mean velocity profiles.
3.10 Figures
Figure 3.1: Solid Model Representation of a Solution Domain
3.10 Figures

Figure 3.2: Test Case 1- Semi-logarithmic Mean Velocity Profiles

Figure 3.3: Test Case 1- Reynolds Shear Stress Profiles
Figure 3.4: Test Case 2- Relaminarising Boundary Layer Experimental Geometries [91]

<table>
<thead>
<tr>
<th>Case</th>
<th>1</th>
<th>2</th>
<th>3 and 4</th>
</tr>
</thead>
<tbody>
<tr>
<td>$l_1$ (mm)</td>
<td>3035</td>
<td>1218</td>
<td>3238</td>
</tr>
<tr>
<td>$l_2$ (mm)</td>
<td>3295</td>
<td>1658</td>
<td>3688</td>
</tr>
<tr>
<td>$l_3$ (mm)</td>
<td>4130</td>
<td>2200</td>
<td>4230</td>
</tr>
<tr>
<td>$d$ (mm)</td>
<td>280</td>
<td>355</td>
<td>355</td>
</tr>
</tbody>
</table>

Figure 3.5: Test Case 2- Inlet Mean Axial Velocity Profiles
Figure 3.6: Test Case 2- Semi-logarithmic Mean Velocity Profiles

Figure 3.7: Test Case 2- Inlet Turbulent Kinetic Energy Profiles
Figure 3.8: Test Case 2- Inlet Reynolds Shear Stress $(\overline{perm})$ Profiles

Figure 3.9: Test Case 2- Contraction Region Mesh
Figure 3.10: Test Case 2- Boundary Layer Region Mesh
Figure 3.11: Comparison of the Axial Development of the Reynolds Number, Shape Parameter, Skin-Friction Coefficient and the Acceleration Parameter for Test Case 2
Figure 3.12: Comparison of the Axial Development of the Non-dimensional Mean Axial Velocity for Test Case 2
Figure 3.13: Comparison of the Axial Development of the Non-dimensional Reynolds Shear Stress \(-\rho u'v'\) for Test Case 2
Chapter 4

Experimental Results

4.1 Introduction

In this chapter the results of the experimental studies are presented. Tests were conducted at two different nozzle scales (48mm (LU48) and 60mm (LU60) nozzle diameters) and across a wide range of NPRs to investigate the effect of Reynolds number and nozzle pressure ratio on the characteristics of nozzle exit boundary layers and subsequent near-field jet plume development.

Section 4.2 describes the measurements made within the nozzle supply duct just upstream of the convergent nozzles (nozzle inlet). These measurements provide a means by which any changes in the state or thickness of the boundary layer that may occur within the nozzle as a result of the rapid acceleration of the flow can be isolated from changes in the conditions supplied to the nozzle itself as operating conditions are varied. Nozzle inlet measurements have typically not been documented in previous studies, but are considered here as important datum states to assess the effect of the imposed nozzle acceleration. Inlet measurements were also used to provide inlet data for the numerical simulations which will be presented in the following chapter.

One of the main objectives of this thesis was to investigate nozzle exit boundary layer changes as influenced by nozzle scale and operating conditions. This was achieved using two measurement techniques and these data are presented and analysed in Section 4.3. Section 4.3.1 presents the nozzle exit boundary layer measurements made using intrusive pneumatic probes; these provide a quick and inexpensive method of boundary layer characterisation but they do not provide turbulence statistics and are limited to subsonic conditions due to the formation of shock waves in front of the probe tip at high subsonic and supersonic nozzle exit.
4.2 Nozzle Inlet Measurements (Pneumatic Probe Data)

velocities. For this reason LDA measurements were also conducted (Section 4.3.2) to allow nozzle exit boundary layer assessment for supercritical nozzle pressure ratios. Before this technique could be used it was important to quantify the influence of the LDA measurement volume length and waist dimensions on the measured profiles as this had been identified as a potential source of error due to velocity gradient broadening, described in Chapter 2. The effect of varying the measurement volume dimensions is described in Section 4.3.2.1 as are the results of adopting different weighting procedures for evaluation of the time-averaged turbulence statistics.

In the literature review in Chapter 2 it was commented that a common practice in nozzle testing is to use a short parallel section at nozzle exit, in order to remove any vena contracta effects due to the inertia of radial inflow velocities caused by the contracting nozzle shape. Warnack and Fernholz [91] have shown that the inner region of highly accelerated boundary layers quickly recovered to a zero pressure gradient-like boundary layer profile once the acceleration had been relaxed. Hence, it is possible that the addition of a parallel nozzle exit section would, in addition to the removal of vena contracta effects, aid boundary layer development by providing a relaxation region in which the boundary layer at the end of the contraction could begin recovery from the effects of the favourable pressure gradient. This might help avoid the relaminarising effects of the convergence. The LDA technique was therefore also used to investigate this by measuring nozzle exit boundary layers in LU48 and LU60 nozzles to which a short parallel extension had been added (LU48(P) and LU60(P)), as described in Chapter 2. The effects of a parallel walled extension are presented in Section 4.4.

Another of the objectives outlined in Chapter 1 was the investigation of the influence of nozzle scale and operating conditions on the development of the near-field of jet plumes. The results of measurements taken to establish the importance of nozzle exit boundary layers on the development of the initial regions of the jet plume for high Reynolds number ($Re_D$), high subsonic and supersonic (underexpanded) jet plumes for convergent and parallel exit section nozzles. The results of these studies are presented in Section 4.5.

4.2 Nozzle Inlet Measurements (Pneumatic Probe Data)

Nozzle inlet measurements were conducted using the delivery pipe boundary layer probe traverse (described in Section 2.2.2) for both nozzle scales and for NPRs ranging from 1.30 to 2.40. Measurements were conducted using the LU48 and LU60 nozzles only, as it was not expected that the addition of any parallel extension would influence delivery duct mass flow rates, static pressures or the development of the boundary layer upstream of the nozzle inlet.
As discussed in the section on data reduction (Section 2.4), all results are presented here in their incompressible (kinematic) forms. This has been shown by Winter and Gaudet [114] as the optimum approach for assessment of boundary layer characteristics since compressibility does not influence the non-dimensional structure of the boundary layer, provided that the free stream Mach number is below \( M \approx 5 \). This allowed easier comparison between results for both nozzle scales and across a wide range of nozzle operating conditions. Results are presented in the form of the global boundary layer parameters \( Re_{\theta_i}, H_{12i}, \theta_i \) and \( c_{f1} \) and mean velocity profiles, non-dimensionalised using both inner and outer wall scaling to assess the state of the near-wall and outer regions of the boundary layer. As direct measurement of wall shear stress was not possible, values of the kinematic friction velocity, \( u_{\tau_i} \), and the local skin friction coefficient, \( c_{f1} \), were deduced using the Clauser plot method which was described in Section 2.4.3.5.

Starting with the inlet boundary layers for the smaller \( LU48 \) nozzle, Figure 4.1 shows the variation of \( Re_{\theta_i}, H_{12i}, \theta_i \) and \( c_{f1} \) with increasing NPR. The boundary layer kinematic momentum thickness Reynolds number shows a steady increase with NPR, rising from \( Re_{\theta_i} \approx 9000 \) to \( Re_{\theta_i} \approx 19000 \) at the highest NPR investigated. Fully turbulent inlet boundary layers are therefore present for all NPRs tested. Despite the large increase in \( Re_{\theta_i} \) with NPR, the kinematic momentum thickness remains nearly constant across the whole range of operating conditions at a value of \( \approx 1.45 \text{mm} \), with corresponding boundary layer thicknesses ranging from \( 21.8 \text{mm} \) at \( NPR \approx 1.30 \) to \( 19.4 \text{mm} \) at \( NPR \approx 2.40 \). Centreline velocities and Mach numbers were not seen to increase significantly with NPR due to the cross sectional area of the delivery pipe being more than 2.5 times that of the nozzle exit area. Core Mach numbers only varied over the range \( 0.230 \leq M \leq 0.276 \). The near linear increase in the momentum thickness Reynolds number is therefore attributed to increases in density at the edge of the boundary layer resulting from increasing delivery duct static pressure with NPR. The kinematic shape factor, \( H_{12i} \), shows clearly turbulent values, starting at 1.341 at \( NPR \approx 1.30 \) and slowly decreasing to 1.327 at \( NPR \approx 2.40 \) as \( Re_{\theta_i} \) increases with NPR. The variation of the local skin friction coefficient with NPR also shows a similar trend to \( H_{12i} \), decreasing with NPR as the velocity profiles become fuller and velocity gradients near the wall decrease.

Figure 4.2 shows the variation of the same global boundary layer parameters with increasing NPR for the \( LU60 \) nozzle. \( Re_{\theta_i} \) shows the same trend as the smaller \( LU48 \) nozzle, but a much wider range, increasing from \( \approx 13000 \) at \( NPR = 1.30 \) to \( \approx 30300 \) at \( NPR = 2.39 \). Kinematic momentum thickness has a similar value to those observed in the \( LU48 \) nozzle starting at 1.42mm at \( NPR = 1.30 \) increasing slightly with NPR until the choked condition \( (NPR = 1.983) \) beyond which \( \theta_i \) begins to decrease slightly. This time the variation in the Reynolds number stems from both increases in velocity and density at the edge of the
boundary layer with NPR. Inlet core Mach numbers for the \textit{LU60} nozzle ranged from 0.322 at \(NPR = 1.30\) to 0.466 at \(NPR = 2.39\). The variation of the shape factor and local skin friction coefficient follow that of the \textit{LU48} nozzle, decreasing as NPR increases. Values are slightly lower, however, due to higher values of \(Re_{\theta i}\) for equivalent NPRs.

At this point a comparison of the data with the results of Winter and Gaudet\cite{114} is included to investigate and demonstrate the benefits of using the kinematic definitions of boundary layer parameters to simplify comparison of results at different operating conditions. Winter and Gaudet suggested several compressibility correction factors which could be used to relate incompressible (kinematic) boundary layer parameters to their compressible forms. Figure 4.3 compares compressible \((H_{12})\) and incompressible \((H_{12i})\) shape factor variation with NPR for the \textit{LU60} nozzle, chosen due to its greater range of core Mach numbers. Also included is the measured kinematic shape factor, corrected using Winter and Gaudet’s compressibility factor \(F_H\), defined as:

\[
F_H = 1 + \frac{M^2}{3} \quad (4.1)
\]

The close agreement between the current measurements processed in compressible boundary layer form \((H_{12})\) and presented in kinematic form and corrected using Equation \((F_H,H_{12i})\) shows that, by applying Winter and Gaudet’s compressibility factor, the compressible shape factor can be recovered in the present experiments from the kinematic shape factor just as was presented in the flat plate data of Winter and Gaudet \cite{114}. In a similar manner, Figure 4.4 compares compressible and incompressible momentum thickness Reynolds number \((Re_\theta\) and \(Re_{\theta i}\)) variation with NPR for the \textit{LU60} nozzle. A compressibility corrected Reynolds number \(F_\delta,Re_{\theta i}\) is also shown where the compressibility factor \(F_\delta\) is defined as:

\[
F_\delta = 1 + 0.056M^2 \quad (4.2)
\]

Again, like the compressibility corrected shape factor, the results show good agreement between corrected \(Re_{\theta i}\) and \(Re_\theta\). Finally, Figure 4.5 shows the variation of the kinematic local skin friction coefficient with \(Re_{\theta i}\) for both nozzles. Also included is the correlation between \(c_{f1}\) and \(Re_{\theta i}\) of Winter and Gaudet:

\[
c_{f1} = \frac{0.01013}{Log10Re_{\theta i} - 1.02} - 0.00075 \quad (4.3)
\]

The data for both nozzles show excellent collapse onto the correlation of Winter and Gaudet. This demonstrates that, in the kinematic form, equivalency exists between the two nozzles, and the current data follows those of the flat plate boundary layer study of Winter and Gaudet. Therefore, the kinematic forms of global boundary layer parameters can be
4.2 Nozzle Inlet Measurements (Pneumatic Probe Data)

used with confidence when comparing boundary layer data for different nozzle operating conditions.

The semi-logarithmic plots Figures 4.6 and 4.7 show the variation of the mean velocity profile for the LU48 nozzle in inner wall units over two ranges, $1.30 < NPR < 1.80$ and $1.80 < NPR < 2.40$. The ranges were selected for ease of presentation. The profiles show a large logarithmic region extending to values of $y^+ \approx 1500$ and a clearly defined wake region. The profiles show excellent agreement with the standard logarithmic law $\left(\frac{1}{0.41}\log(y^+) + 5.2\right)$ across the whole range of NPRs. Close inspection of the profiles reveals small increases in the extent of the logarithmic region which is expected as the logarithmic region grows with momentum thickness Reynolds number [29], which has been shown to increase with NPR. Similar plots for the LU60 nozzle (Figures 4.8 and 4.9) also show good agreement between profiles, the only difference being the greater extent of the logarithmic region and a more pronounced wake region due to the greater values of $Re_{hi}$ in the delivery pipe upstream for the LU60 nozzle compared to the LU48 nozzle.

Presenting the same results but scaled using outer wall units shows the development of the outer region of the boundary layer. For the LU48 tests, Figures 4.10 and 4.11 (and similar plots for the LU60 nozzle, Figures 4.12 and 4.13) show the mean velocity plotted in terms of the velocity defect, $\frac{(U_\delta - U)}{u_r}$, against wall distance non-dimensionalised with respect to the Rotta-Clauser length, $\Delta$, defined as:

$$\Delta = \int_0^\infty \frac{U_\delta - U}{u_r} \, dy$$

(4.4)

Where $U_\delta$ is defined as $0.995U_{CL}$. Also shown in the figures is the universal logarithmic velocity distribution for outer law velocity profiles given by Fernholz and Finley[67]:

$$\frac{U_\delta - U}{u_r} = -4.70 \ln \left(\frac{y}{\Delta}\right) - 6.74$$

(4.5)

The profiles collapse well for both nozzles and for all NPRs investigated. The outer edge ($\frac{\Delta}{x} > 5 \times 10^{-3}$) also shows good agreement with the linear behaviour of the universal velocity distribution for zero pressure gradient boundary layers [67]. There is a small degree of scatter particularly in the LU60 data toward the lower end of the scale which is attributed to differing extents of the inner wall logarithmic region, as highlighted in Fernholz and Finley [67].

Sreenivasan [71] argued that it was essential to have fully developed turbulent boundary layers when studying the effects of pressure gradients on boundary layer development. As one of the main purposes of this thesis was to investigate the effects of propulsion nozzle geometry, and therefore the effects of the static pressure gradients on boundary layer development, fully developed boundary layers in the approach flow were clearly desirable. The
inlet measurements have shown that, for both nozzle scales and all operating conditions under investigation, the boundary layers prior to entering the nozzle are fully turbulent, with developed near-wall and outer wall regions which are similar to high Reynolds number, zero pressure gradient boundary layers and, therefore, meet the requirements of Sreenivasan.

Another important observation to be made here is the ratio of the mass flux contained within the boundary layer at the nozzle inlet compared to the total mass flux of the fluid supplied to the nozzle. As mentioned in Chapter 2, any fluid that has been contained within a boundary layer will have a total pressure less than that of the freestream due to viscous losses. At least the same proportion of the fluid at nozzle exit will also have a total pressure lower than that of the freestream. This has important implications when considering the nozzle exit measurements using pneumatic probes. As detailed in Chapter 2, nozzle exit velocity profiles derived from pneumatic probe measurements use gas dynamic relationships which relate local total pressure and wall static pressure to local velocity. Although the boundary layer (defined as the viscous effected region of the flow in proximity to the nozzle wall) is expected to become thinner due to the effects of the favourable pressure gradient, the total pressure (used to derive boundary layer profiles) at the edge of this region may continue to increase towards the centreline even though the region beyond the edge of the boundary is essentially inviscid which introduces uncertainty when defining the edge of the boundary layer. An indication of the mass flux within the boundary layer at nozzle inlet can be deduced from the integral:

\[
\dot{m}_b = 2\pi \int_{R-\delta}^{R} \rho U r dr
\]  

The total mass flow supplied to the nozzle is given by:

\[
\dot{m} = 2\pi \int_{0}^{R} \rho U r dr
\]

Where \( \delta \) is the radial location of the boundary layer edge corresponding to \( U_\delta = 0.995 U_{CL} \). Evaluation of the ratio \( \frac{\dot{m}_b}{\dot{m}} \) reveals that, for all operating conditions, around 80% of the fluid entering the nozzle has experienced some viscous losses and will have a total pressure lower than that of the centreline. Therefore at least 80% of the fluid at nozzle exit will also have a total pressure lower than the centreline value. If the nozzle exit boundary layer has become thinner due to the acceleration it will be contained within regions of the flow with total pressure lower than the centreline values and therefore measured total pressures may continue to grow beyond the edge of the "true" boundary layer at the nozzle exit.

Finally, although no turbulence measurements have been made as part of the of the present nozzle inlet data, the closeness of the mean velocity profiles to standard zero pressure gradient profile shapes measured in many previous boundary layer studies means that...
previous data from such studies on turbulence quantities (for example, Fernholz and Finley [67]) could be used with confidence if, for example, the present measurements needed to be supplemented with turbulence quantities for CFD inlet condition purposes.

4.3 Nozzle Exit Measurements

4.3.1 Pneumatic Probe Data

The following section details the results of the intrusive pneumatic probe nozzle exit boundary layer measurements; the same approach to quantifying the state and shape of the exit boundary layer was used as adopted in the work of Lepikovsky et al. [5], who detailed nozzle exit variation for high speed and high temperature flows. Operating conditions were restricted to subcritical NPRs as the static pressure assumptions used during data reduction (described in Section 2.4) become invalid once the critical NPR is reached and the nozzle chokes. Results are presented for both LU48 and LU60 nozzles in terms of global boundary layer parameters $Re_o$, $\theta_i$, $H_{12i}$, and non-dimensional mean velocity profiles using the methods and assumptions detailed in Chapter 2.

Figure 4.14 shows the variation of the kinematic momentum thickness Reynolds number with nozzle pressure ratio for the LU48 nozzle. $Re_{\theta_i}$ increases gradually from $Re_{\theta_i} \approx 480$ at $NPR \approx 1.30$ to $Re_{\theta_i} \approx 720$ at $NPR \approx 1.90$. The rate at which the Reynolds number increases with NPR decreases as the choked condition is approached where the data flatten out, which is noticeably different to the behaviour of the inlet boundary layers whose kinematic momentum thickness Reynolds numbers continue to increase linearly beyond the choked condition thus suggesting the severity of the acceleration increases with NPR. The effects of the acceleration on the boundary layer within the nozzle are immediately apparent, since $Re_{\theta_i}$ has decreased from $Re_{\theta_i} \approx 9000$ and $Re_{\theta_i} \approx 17000$ at $NPR \approx 1.30$ and $NPR \approx 1.90$ to $\approx 480$ and $\approx 720$. This lower values remain above the critical value for turbulence suggested by Fernholz and Finley [67] but represents a change of a factor of about 20, despite the large increase in the velocity at the edge of the boundary layer due to acceleration (recalling that $M_{inlet} = 0.276$ at the point at which the nozzle chokes and $M_{exit} = 1$). The reduction in the $Re_{\theta_i}$ stems from the dominance of the thinning of the boundary layer due to the acceleration over the increase in the velocity and density of the fluid at the edge of the boundary layer. This is highlighted by the variation of kinematic momentum thickness, $\theta_i$, with NPR presented in Figure 4.15. The data show a gradual decrease in momentum thickness with NPR, with values starting at around 0.034mm and corresponding boundary layer thickness($\delta$) $\approx 0.5mm$, decreasing to $\approx 0.024mm$ ($\delta \approx 0.375mm$). The degree of scatter in the data is primarily associated with errors arising from the interpolation needed to
determine the effective wall location. Comparing the results with those of the inlet boundary layers in Figure 4.1 highlights the effect of acceleration as \( \theta_i \) has been reduced by more than 20 times through the nozzle.

The shape (and hence by implication, the state) of the boundary layer has also been significantly altered during its passage through the nozzle; this is shown by the variation of the kinematic shape factor, \( H_{12i} \), presented in Figure 4.16. Despite considerable scatter due to interpolation, \( H_{12i} \) reveals a transitional boundary layer shape which becomes more laminar-like as \( \text{NPR} \) increases, with values ranging from \( H_{12i} \approx 2 \) at \( \text{NPR} \approx 1.30 \) to \( H_{12i} \approx 2.30 \) at \( \text{NPR} \approx 1.90 \), compared to the highly turbulent values (in the region of 1.35) across the whole range of \( \text{NPRs} \) measured upstream of nozzle inlet. At this point it is worth noting that care is needed when interpreting results at near-critical pressure ratios as it is not known to what extent the shock wave formation that will result even at sub-critical \( \text{NPRs} \) due to local acceleration of the flow over the probe’s sensing tip influences the measurements and the boundary layer itself, and some of the scatter is undoubtedly due to this.

Turning attention to the mean velocity profiles, Figures 4.17 and 4.18 show the variation of the mean velocity profiles, non-dimensionalised with respect to \( U_b \) and the kinematic displacement thickness, \( \delta^*_b \). \( \delta^*_b \) was used as the reference length as the boundary layer thickness was difficult to define due the growth of the total pressure toward the centreline for reasons described in the previous section. \( \delta^*_b \) was insensitive to the location of the boundary layer edge. Also included in these figures is the profile upstream of the nozzle inlet for \( \text{NPR} = 1.894 \) for comparison. Inspection of the nozzle exit profiles shows that there is very little variation across the whole \( \text{NPR} \) range, particularly for distances below \( y/\delta^*_b \approx 1.5 \) where the profiles collapse perfectly. Close inspection does show a small variation in the region \( 1.5 < y/\delta^*_b < 5 \) where the profiles become flatter with increasing \( \text{NPR} \). The reasons for the laminar-like values of the kinematic shape factor become apparent when the exit profiles are compared with the representative, fully turbulent, inlet profile. The boundary layer profiles at inlet and exit show very significant differences, velocity profiles at the nozzle exit show a more gradual, linear increase between the wall and \( y/\delta^*_b \approx 2 \) followed by an overshoot of the turbulent profile as expected when contrasting laminar-like and fully turbulent boundary layers.

To identify the effects of the favourable pressure gradient on the boundary layer it is preferable to present the profiles appropriately scaled to identify the key regions of the boundary layer, such as semi-logarithmic plots of the near-wall region. Unfortunately, due to some uncertainty in determining the wall location in the exit probe data and due to the highly suppressed nature of the exit profiles, it was not possible to determine a precise value of the wall friction velocity at nozzle exit. Instead, the profiles are presented in the
form of the semi-logarithmic Clauser plot. A similar approach was adopted by Launder in his study of relaminarisation [75]. Using such a plot means the slope of any logarithmic region varies with the local skin friction coefficient, but if the boundary layer does contain a true logarithmic region the data will collapse on to a line of constant $c_{f1}$ in this region. A Clauser plot of the nozzle exit profiles is shown in Figure 4.19. Also included are two profiles from upstream of the nozzle inlet corresponding to $NPR = 1.341$ and 1.816 for comparison purposes. Various lines of constant $c_{f1}$ are also presented for visualisation purposes.

The reference inlet profiles show clearly defined logarithmic and wake regions with $c_{f1}$ values of just less than 0.003. At the nozzle exit the profiles are seen to lie well above the inlet profiles (denoting high local skin friction coefficients and velocity gradients at the wall). This appears to contradict the velocity profiles shown in Figures 4.17 and 4.18, where velocity gradients appear to be greater at inlet. This is due to the profiles being non-dimensionalised with respect to $\delta_i^*$ which is approximately a factor of 20 greater at inlet compared to at exit. There is nothing that could be identified as a wake region, which is to be expected as it is the outer edge of the boundary layer that will be most affected by the onset of a large favourable pressure gradient [77]. It may be argued that a logarithmic region still exists as a small extent of the profiles can still be seen to be parallel to the lines of constant $c_{f1}$, but the extent of this region has been significantly reduced. The nozzle exit boundary layers are clearly no longer in equilibrium. Comparing the nozzle exit profiles at the varying NPRs it can be seen that the profiles agree closely, suggesting that the effects of the acceleration are similar across the whole range of NPRs.

Attention is now turned to the results of nozzle exit measurements for the larger LU60 nozzle. The variation of $Re_{\theta_i}$ with NPR in Figure 4.20 again shows large decreases in $Re_{\theta_i}$ magnitude compared to the inlet values (Figure 4.2). Magnitudes of $Re_{\theta_i}$ at the nozzle exit are similar to those of the smaller LU48 nozzle with values ranging from $\approx 550$ to around $\approx 750$ at the point at which the nozzle chokes. The reduction in $Re_{\theta_i}$ is now of order 40, almost double that observed in the results of the LU48 nozzle. The variation of $\theta_i$ is also more pronounced compared to the LU48 nozzle with a reduction of order 50 compared to the inlet boundary layer. This shows that there are significant differences in the effects of the acceleration of the boundary layers within the two different nozzle scales which can be attributed to the short contraction length and hence more rapid acceleration of the boundary layer within the large LU60 nozzle.

Despite the apparent variations in the development of the boundary layer within the nozzle, values at the nozzle exit appear remarkably similar for the two scales. This trend is continued in the variation of $H_{12i}$ with NPR, which is presented in Figure 4.22. Again, transitional values in the region of $1.9 - 2.0$ are seen at the lower end of the NPR range.
4.3 Nozzle Exit Measurements

which increase with NPR. Despite considerable scatter, distinct trends can be seen in the data. Unlike the LU48 results which show a gradual rise with NPR, the LU60 results show a two-stage variation, remaining fairly constant up to $NPR \approx 1.65$, beyond which value begins to grow to $H_{12i} \approx 2.2$ at $NPR \approx 1.90$. At present the reason for this behaviour is not fully clear as, at this point, the flow within the nozzle is not known. The results do suggest that the magnitude of the acceleration in both the LU48 and LU60 nozzles may reach values at which relaminarisation may occur at NPRs above 1.65.

Similar agreement between the two nozzle scales is seen in the non-dimensional mean profiles in Figures 4.23 and 4.24. Again the exit boundary layer profiles are flatter compared to the fully turbulent inlet profile provided and show close agreement in the region $0 \leq y/\delta_i < 1.5$. Like the LU48 profiles, the velocity profiles can be seen to flatten out with increasing NPR in the region $1.5 \leq y/\delta_i < 7$; for the LU60 nozzle, greater variation is seen in this region which, again, suggests greater acceleration of the boundary layer within the LU60 nozzle.

Presenting the mean velocity profiles in the form of Clauser plots (Figure 4.25) shows similar departure from the equilibrium state as those observed for the LU48 data with the disappearance of a defined wake region and departure from the logarithmic law of the wall. Unlike the results for the LU48 nozzle, the profiles can be seen to consistently move toward higher local skin friction coefficients with increasing NPR. The presence of a logarithmic region is less clear, but the profiles still appear to be turbulent as the velocity profiles still flatten out abruptly beyond the location corresponding to $U_2 \approx 0.90$ which was not present in the Clauser plots of relaminarised boundary layers presented by Launder [75].

The inlet and exit boundary layer data have shown that the acceleration of the boundary layers within the nozzle can significantly influence the nozzle exit boundary layer reducing values of $Re_{\theta i}$ by more than an order of magnitude and change the shape from fully turbulent to laminar-like over short axial distances. Turning attention to the results of Lepicovsky [5], the present data give further weight to the observations in Trumper [62] that the low $Re_{\theta i}$, laminar-like exit boundary layers of Lepicovsky may be due to the effects of the acceleration (relaminarisation) through the nozzle (assuming a turbulent inlet boundary layer) and not necessarily a result of natural transition as suggested by Lepicovsky.

Before drawing definite conclusions on the effects of favourable pressure gradients and NPR on the state and shape of nozzle exit boundary layers, a second, non-obtrusive measurement technique was used to measure the nozzle exit boundary layers. The use of the LDA measurement technique provided an independent set of results which could be also gathered for supercritical operating conditions as well as valuable information on turbulence statistics that cannot be obtained using pneumatic probes. The LDA data are discussed in
4.3 Nozzle Exit Measurements

the following section.

4.3.2 LDA Data

4.3.2.1 Influence of LDA Measurement Volume

Before using this technique, it was important to consider the influence of the measurement volume size on the measured mean velocities and turbulence statistics given the small dimension. In the review of error sources in LDA measurements in Chapter 2 the effect of the measurement volume size on measured mean velocities and turbulence statistics has been discussed. The analyses of Durst et al. [153] and Compton [130] showed the effect of velocity gradient broadening that results from the presence of non-uniform velocity gradients across the finite measurement volume formed at the intersection of the LDA beam pairs (particularly when measuring in the near-wall region of boundary layers where velocity gradients are large). Their analyses assumed that the only significant gradient existed across the waist (minor axis) of the measurement volume, which is the case when measuring 2D planar boundary layers. The same cannot be assumed when measuring thin axisymmetric boundary layers or the initial stages of round jet shear layers, where the azimuthal curvature of the boundary layer or shear layer will impose additional gradients along the length of the LDA measurement volume (as illustrated in Figure 2.25).

For this reason, a study of the influence of the measurement volume size on the data taken in the nozzle exit boundary layers was performed to assess the influence of the measurement volume dimensions. Three LDA configurations were used. The first configuration (referred to below as the datum case) was without beam expanders, and relevant parameters for this case were shown in Table 2.1 in Chapter 2 (as a reminder, the axial length of the measurement volume was approximately 2.4mm with dimensions of the other two orthogonal directions of approximately 0.15mm). The second configuration used beam expanders and reduced beam separation which led to roughly the same maximum bandwidth but half the major axis and minor axis lengths when compared to the datum (parameters shown in Table 2.2 in Chapter 2). The reduced beam separation was required to increase the fringe spacing without increasing the waist of the measurement volume to ensure that an adequate bandwidth was available to resolve the greatest instantaneous velocities expected. A third configuration was tested, with the same measurement volume waist dimensions as configuration 2, but with a further reduction of the measurement volume length to approximately 0.6mm. In practice this configuration could not be used to perform measurements for all NPRs due to limited bandwidth, but it was included in this study to allow further assessment of the influence of the measurement volume length. Measurement volume characteristics for this configuration
are shown below in Table 4.1. A low NPR of 1.25 was used in all tests so that the nozzle whose exit velocity was such \( U_{exit} \approx 170m/s \) that all three configurations could be used. Results from the three configurations are presented in terms of mean velocity profiles (Figure 4.26), Clauser plots (Figure 4.27) and the RMS of the mean axial velocity (Figure 4.28). In the figures 38\( mm, E = 1.00 \) (where \( E \) is the beam expansion ratio) denotes the datum case without beam expanders, 20\( mm, E = 1.98 \) denotes the case with beam expanders but the same measurement volume length and 38\( mm, E = 1.98 \) denotes beam expanders and reduced measurement volume length.

<table>
<thead>
<tr>
<th></th>
<th>( U_{BSA1} )</th>
<th>( U_{BSA2} )</th>
</tr>
</thead>
<tbody>
<tr>
<td>Wavelength (nm)</td>
<td>514.5</td>
<td>488</td>
</tr>
<tr>
<td>Focal Length (mm)</td>
<td>310</td>
<td>310</td>
</tr>
<tr>
<td>Beam Spacing (mm)</td>
<td>38</td>
<td>38</td>
</tr>
<tr>
<td>Expansion Ratio</td>
<td>1.98</td>
<td>1.98</td>
</tr>
<tr>
<td>Fringe Spacing (( \mu m ))</td>
<td>2.135</td>
<td>2.025</td>
</tr>
<tr>
<td>Number of Fringes</td>
<td>35</td>
<td>35</td>
</tr>
<tr>
<td>Measurement Volume Diameter (mm)</td>
<td>0.076</td>
<td>0.072</td>
</tr>
<tr>
<td>Measurement Volume Length (mm)</td>
<td>0.631</td>
<td>0.598</td>
</tr>
</tbody>
</table>

Table 4.1: Measurement Volume Characteristics With Beam Expanders and 38\( mm \) separation

The profiles of mean velocity, non-dimensionalised with respect to the velocity at the edge of the boundary layer, \( U_\delta \), and kinematic displacement thickness, \( \delta^*_\| \), showed an influence of the measurement volume on the profile in the near-wall region. At the outer edge of the boundary layer, where the velocity gradients are small, all three of the profiles show good agreement. Moving closer to the wall, the profile without the beam expanders can be seen to overshoot compared to the results where beam expanders were used in the region \( 2 < \delta^*_\| < 7 \). This is followed by an undershoot below \( \delta^*_\| \approx 2 \). Comparing the two cases with beam expanders (identical measurement volume waist diameters but different lengths) little variation between the results is apparent, particularly in the region below \( \delta^*_\| \approx 7 \) where velocity gradients are at their greatest. The greatest deviation between the profiles occurs at \( \delta^*_\| \approx 1 \) which equates to the edge of the buffer region. The study by Durst et al. [124] also found that the greatest deviation corresponded to the edge of the buffer region where the second derivative \( \frac{d^2U}{d\delta^*_\|} \) was at its greatest. The lower measured velocities for the case without beam expanders below \( \delta^*_\| \approx 2 \) is made more apparent in the Clauser plot of Figure 4.27. Again, the profiles show excellent agreement between both beam expander cases; the very slight difference at the lower end of the scale is attributed to the error introduced due to the interpolation of the profiles to establish the location of the wall, which was more difficult.
4.3 Nozzle Exit Measurements

with the greater beam separation as the measurement volume was located approximately twice as far downstream of the nozzle exit compared to when 20mm beam spacing was used (to avoid beam interference with the nozzle wall). The results of the case without the beam expanders show values as much as 25% lower than with beam expanders.

The influence of the measurement volume size is also apparent in the plot of RMS velocity profiles, non-dimensionalised with respect to the velocity at the edge of the boundary layer and kinematic displacement thickness, in Figure 4.28. All three profiles collapse onto each other for distances above $3\delta^*$, where velocity gradients tend to zero. Large variations begin to appear close to the wall as the velocity gradient increases and the range of velocities covered by the measurement volume increases. Little variation can be seen between both cases with beam expanders, but RMS values of the case without overshoots the other profiles, with a peak value of 0.20 compared to 0.175, an overshoot of 15%.

The results clearly show the influence of the lengths of the minor and major axes on the measured mean and RMS velocities. Large variations in measured values of the mean velocities occur when the waist of the measurement is halved, whereas no obvious variation was seen when waist was kept constant and the length of the measurement volume was halved. This shows that the velocity gradients across the waist are more influential on the measured boundary layer profiles than the gradients imposed along the length of the measurement volume due to boundary layer or shear layer curvature and, therefore, the compromise between measurement volume length and the requirement of large bandwidth is not significant. Since the influence of measurement volume on the measured mean and turbulence statistics have now been established, the LDA technique is now used to measure nozzle exit boundary layers, the results of which are presented in the following section. For all subsequent measurements, the beam expander configuration used in case 2 was adopted.

4.3.2.2 Influence of Weighting Method

As described in Chapter 2, transit time weighting was employed during processing of the LDA data to produce time-averaged quantities. Transit time weighting was adopted in order to reduce the effects of velocity bias on time-averaged data, which was identified as a possible source of error in Chapter 2. In this section the influence of the choice of weighting method on profiles of mean velocity and turbulence statistics and on global boundary layer parameters is investigated. Figure 4.29 compares the non-dimensional mean velocity profiles with and without transit time weighting on the profiles measured in the previous section. The profiles show that, despite differences between dimensional data, when non-dimensionalised with respect to the kinematic displacement thickness the profiles collapse well. This indicates that the choice of transit time weighting has little influence on the mean profile. The same is true
4.3 Nozzle Exit Measurements

of the turbulence statistics. Figures 4.30 and 4.31 compare weighted and unweighted non-dimensional profiles of RMS axial velocity and Reynolds shear stress $\overline{u'v'}$. Good agreement between RMS profiles is observed with peak values of the same magnitude occurring at the same non-dimensional distance from the wall. The same is seen in the profiles of Reynolds shear stress in Figure 4.31. The outer regions collapse and the variation in peak magnitudes are small (the unweighted peak is 3% larger than in the weighted data). Figures 4.32 and 4.33 show the effect of the weighting method on the $H_{12i}$ and $Re_{\theta i}$ for the nozzle exit boundary layers of the LUGO nozzles. The figures show little difference between weighted and unweighted results. Most importantly the variation of the data with NPR (which will be discussed in detail in the following section) appear to be identical indicating that any observed trends are not artifacts introduced by the choice of weighting method.

4.3.2.3 LDA Results

LDA measurements at nozzle exit are now presented, compared and contrasted with the relevant pneumatic probe measurements.

Figure 4.34 shows the variation of $Re_{\theta i}$ with NPR for the LDA and probe data. $Re_{\theta i}$ increases gradually with NPR in a similar fashion to the pneumatic probe measurements in Figure 4.14. Despite scatter in the data at the higher NPRs, $Re_{\theta i}$ is observed to continue to increase beyond the point at which the nozzle chokes which could not be established using the probe. When the magnitudes of $Re_{\theta i}$ are compared with the probe results for the subcritical NPRs, the LDA results are consistently lower ($440 < Re_{\theta i} < 600$ (LDA) compared to $550 < Re_{\theta i} < 750$ (probe) for the range $1.30 < NPR < 1.90$) but still show the very large reduction in $Re_{\theta i}$ through the nozzle. Figure 4.35 presents the variation of $\theta_i$ with NPR. The LDA data again show the same decrease with NPR but lie consistently below the probe data. This is consistent with the observations of $Re_{\theta i}$ above. The decrease with NPR is seen to continue in the LDA data as the critical NPR is exceeded following the same trend line. Variation between probe and LDA data is most likely a result of the constant static pressure assumption which was used when deriving velocity from measured total pressure as radial variation of static pressure could not be measured directly due to the small thickness of the boundary layer. If the static pressure increases towards the centreline the velocities derived using a constant static pressure will be greater, resulting in a fuller profile and larger momentum thickness. The variation of $H_{12i}$ with NPR, presented in Figure 4.36, compares reasonably well with the probe data, with values in the region of 2.1, but the LDA results do not show the shift toward higher, more laminar-like values with increasing NPR. The departure of the LDA and probe data and increased scatter in the probe data as the critical NPR is approached indicates possible probe interference effects at the higher NPRs due to
local shock wave formation in front of the probe sensing tip. This will not appear in the LDA results.

The LDA results show an increase in $H_{12i}$ beyond the critical NPR. For supercritical NPRs the jet plume enters the underexpanded regime and a Prandtl-Meyer expansion fan will form and emanate from the nozzle lip as the flow is forced to accelerate to achieve equilibrium with ambient conditions (discussed in more detail in Section 4.5.2, below). As the boundary layer measurements were actually made a small distance downstream of the nozzle exit (approximately 15 exit boundary layer momentum thicknesses), the traverse line will intersect the expansion fan. The apparent increase in the shape factor to $H_{12i} \approx 2.30$ at $NPR \approx 2.4$ may be the result of the superposition of the velocity profile across the expansion fan onto the boundary layer velocity profile. The non-dimensional mean velocity profiles presented in Figures 4.37 and 4.38 demonstrate good agreement between profiles for all of the subcritical NPRs but show the profiles becoming more flattened and laminar-like in the region $1.5 < y/\delta_i < 7$ once the nozzle has choked, which reflects the changes noted in $H_{12i}$ above. Figure 4.39 shows a comparison between probe and LDA mean velocity profiles presented non-dimensionally as Clauser plots. The figure shows that both measurement techniques follow the same trends, namely the disappearance of clear logarithmic and wake regions (present in the inlet boundary layer profiles) and an overshoot of the inlet profiles toward higher values of $C_f$. Agreement between Probe and LDA measurements for the LU60 nozzle mean velocity profile and global boundary layer parameters were similar to the LU48 nozzles and are not discussed.

Figures 4.40 and 4.41 show the variation of the boundary layer RMS velocity profiles, non-dimensionalised with respect to $U_\delta$, for the LU48 nozzle. The profiles show the peak value in the near-wall region increasing with NPR, rising from $0.17U_\delta$ at $NPR \approx 1.30$ to $0.25U_\delta$ at $NPR \approx 2.40$. The increase in peak magnitude with NPR is also shown in Figure 4.42 which shows the peak value to increase linearly with NPR up to the critical NPR, beyond which it remains constant. Turbulence levels at the outer edge of the boundary layer do not appear to be affected by increases in NPR, remaining around $0.02U_\delta$. Warnack and Fernholz [91] and Sreenivasan [71] observed increases in near-wall $(y/\delta < 0.1)$ turbulence levels in highly accelerated (laminarreiscent and relaminarising) boundary layers as the turbulence responded to increased $u_\tau$ as the velocity profile becomes flatter. The increasing magnitude indicates the severity of the acceleration increases with NPR. Warnack and Fernholz [91] also observed near constant turbulence levels in the outer region of the boundary layer when the flow was accelerated. Additional changes in the profiles are seen in the central region of the boundary layer $(2 < y/\delta_i < 8)$ where turbulence levels continue to decrease with increasing NPR, resulting in a flatter profile. The same decrease in turbulence levels in the central region of the boundary layer was also observed by Warnack and Fernholz [91]. The turbulence
4.3 Nozzle Exit Measurements

Profiles for the larger $LU_{60}$ nozzle are shown in Figures 4.43 and 4.44. Like the results of the $LU_{48}$ nozzle, levels in the central region decrease and near-wall peak values increase with increasing NPR. The change in the peak values (Figure 4.42) are virtually identical those observed in the $LU_{48}$ nozzle up to the critical NPR, but are considerably higher (up to $0.31U_A$) beyond the choked condition which is almost certainly a result of the traverse line crossing the expansion fan emanating from the nozzle lip. Variations between $LU_{48}$ and $LU_{60}$ are likely to be a result of the different axial location of the measurement traverse line relative to the expansion fan.

Figures 4.45 and 4.46 show the variation of profiles of the Reynolds shear stress $u'v'$, non-dimensionalised with respect to $U_A^2$, with NPR for the $LU_{48}$ nozzle. The profiles for the subcritical pressure ratios collapse well above $y/\delta_* \approx 1$ but peak values appear to increase initially with NPR, reaching a maximum of $-\frac{u'v'}{U_A^2} \approx 0.004$ at $NPR = 1.387$ then decreasing slowly to $-\frac{u'v'}{U_A^2} \approx 0.0015$ at $NPR = 1.864$. Peculiar behaviour is observed beyond the critical NPR. Peak values (shown in Figure 4.47) rapidly decrease with NPR and by $NPR = 2.173$ the whole of the profile becomes negative. The variation of $u'v'$ with NPR for the $LU_{60}$ nozzle is shown in Figures 4.48 and 4.49. Like those of the smaller $LU_{48}$ nozzle, the Reynolds shear stress in the outer region of the boundary layer collapse well but peak values now continually increase with NPR, starting around $-\frac{u'v'}{U_A^2} \approx 0.004$ at the lower end of the NPR range and reaching a maximum of $-\frac{u'v'}{U_A^2} \approx 0.0115$ at $NPR = 2.241$ which greatly exceeds the peak values seen in the results of the smaller nozzle. Beyond this NPR peak values begin to decrease rapidly, reaching $\frac{u'v'}{U_A^2} \approx 0.006$ at the highest NPR investigated. Although the profiles for the $LU_{60}$ nozzle do not exhibit the same change of sign for supercritical NPRs as seen in the results of the $LU_{48}$ nozzle, large decreases in the magnitude of the shear stress in the near-wall region are similar to those seen in the smaller nozzle. This behaviour is likely to be the result of the superposition of the Prandtl-Meyer expansion fan that originates from the nozzle lip when operating at underexpanded conditions on the boundary layer profile. The position of the traverse line ($LU_{60}$ nozzle) relative to the expansion fan is illustrated in Figure 4.50 (for $NPR = 2.40$). In the results for the $LU_{48}$ nozzle, where the magnitude of the Reynolds shear stress are considerably lower than for the $LU_{60}$ nozzle, the effects of superimposing the expansion fan on the boundary layer profile may be of sufficient magnitude to cause the observed inversion. The effects of the expansion fan on the measured profiles is more clearly illustrated in the following section.
4.4 Influence of Addition of a Parallel-Walled Nozzle Exit Extension

The study of Warnack and Fernholz [91] showed that for a laminarescent boundary layer (still turbulent but exhibiting laminar-like characteristics) the near-wall region of the boundary layer quickly recovered from the effects of a large favourable pressure gradient once the acceleration was relaxed, with near-wall profiles that closely resembled the canonical zero pressure gradient turbulent boundary layer reappearing quickly. In light of this, parallel-walled extension pieces were added to the end of each of the contraction nozzles (denoted LU48(P) and LU60(P)) to provide a region within the nozzle in which the acceleration was relaxed, allowing some distance over which the boundary layer could recover prior to leaving the nozzle to form the initial jet free shear layer. The LU48(P) featured a 34.12mm parallel section and the LU60(P) 31.84mm, as shown in Figure 2.6. As the nozzle exit boundary layer behaviour for the contraction-only LU48 and LU60 nozzles was very similar over the whole NPR range it was decided that a programme of boundary layer measurements for both nozzles and across the whole NPR range was not required; instead 3 conditions were selected, two subsonic conditions, \( \text{NPR} = 1.50 \) and \( \text{NPR} = 1.89 \) (on the point of choking) and a supersonic, underexpanded case with \( \text{NPR} = 2.40 \). Results are again presented in terms of boundary layer parameters \( \text{Re}_{\text{oi}}, \theta_i, H_{12i} \) as well as profiles of the mean velocity and turbulence statistics.

Figure 4.51 shows the mean velocity profiles at the exit of the LU60(P) nozzle. Velocities have been non-dimensionalised by \( U_{CL} \). Attention is directed to the profile in the underexpanded case. Moving away from the wall location the velocity is seen to increase to a value of \( 1.07U_{CL} \) then decrease toward the centreline. The bulge in the profile is introduced by the presence of a Prandtl-Meyer expansion fan originating from the nozzle lip.

Table 4.2 shows the nozzle exit boundary layer parameters for the two subsonic cases and for both nozzles. It is clear that, for both nozzles, the addition of the parallel extension has significantly altered the thickness and the shape of the nozzle exit boundary layers compared to the contraction-only nozzle geometry. Values of \( \text{Re}_{\text{oi}} \) have increased significantly to fully turbulent values. The greatest increase occurred for the LU60(P) nozzle, from \( \text{Re}_{\text{oi}} \approx 440 \) and \( \text{Re}_{\text{oi}} \approx 430 \) at \( \text{NPR} \approx 1.5 \) and \( \text{NPR} \approx 1.88 \) (LU60) to \( \text{Re}_{\text{oi}} = 7494 \) and \( \text{Re}_{\text{oi}} = 6852 \) (LU60(P)) representing more than an order of magnitude change. The shape of the boundary layer has also significantly changed as indicated by the decrease in \( H_{12i} \) from laminar-like values in the region of 2.0 – 2.1 for both LU48 and LU60 nozzles to the clearly turbulent values of 1.22 – 1.43 for the LU48(P) and LU60(P) nozzles. Also apparent is a difference in the effect of adding the parallel-walled extensions to the two different diameter nozzles.
Recalling that the thickness and shape of the nozzle exit boundary layers for both LU48 and LU60 nozzles were shown to be similar in the previous section, the results with a parallel-walled extension show a greater recovery for the larger LU60(P) nozzle with values of $Re_{oi}$, $\theta_i$, and $H_{12i}$ being approximately twice that achieved in the smaller LU48(P) nozzle. This is despite the fact that the length of the parallel section in which the boundary layer has chance to recover in the smaller nozzle is greater than that of the larger nozzle (34.12mm, compared to 31.84mm).

![Table 4.2: Boundary Layer Parameters for Nozzle With Parallel Walled Extension](image)

<table>
<thead>
<tr>
<th></th>
<th>NPR 1.50</th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>LU48(P)</td>
<td>LU48</td>
<td>LU60(P)</td>
<td>LU60</td>
<td>LU48(P)</td>
<td>LU48</td>
<td>LU60(P)</td>
</tr>
<tr>
<td>$\theta_i (\times 10^{-4} m)$</td>
<td>1.54</td>
<td>0.258</td>
<td>3.62</td>
<td>0.248</td>
<td>1.68</td>
<td>0.216</td>
<td>2.39</td>
</tr>
<tr>
<td>$H_{12i}$</td>
<td>1.43</td>
<td>2.11</td>
<td>1.22</td>
<td>2.00</td>
<td>1.33</td>
<td>2.15</td>
<td>1.22</td>
</tr>
<tr>
<td>$Re_{oi}$</td>
<td>3213</td>
<td>495</td>
<td>7494</td>
<td>442</td>
<td>3513</td>
<td>525</td>
<td>6852</td>
</tr>
</tbody>
</table>

Figures 4.52 and 4.53 show the non-dimensional mean velocity profiles for both subsonic NPRs for the LU48(P) and LU60(P) nozzles, respectively. Compared to the contraction only geometry the profiles show a fuller, more turbulent distribution in the region $y/\delta_i < 5$. Comparing the results of the LU48(P) and LU60(P) nozzles the greater recovery of the larger of the two nozzles is clearly apparent with a fuller profile demonstrated by the LU60(P) nozzle.

The recovery of the boundary layer to a fully turbulent state is better illustrated when plotted in a semi-logarithmic form. Like the contraction-only nozzle exit boundary layers, the mean profiles are presented in the form of Clauser plots in Figures 4.54 (LU48(P)) and 4.55 (LU60(P)). Also included are the contraction-only profiles to highlight the changes. For both nozzles, a logarithmic region has reappeared in the near-wall region ($\approx 3000 < u' \delta_i < \approx 20000$ for both nozzles). Unlike the inlet boundary layers, the profiles do not follow the lines of constant $c_f$, instead the gradients of the logarithmic regions are considerably higher which indicates the presence of an adverse pressure gradient within the nozzle [64]. For both nozzles and NPRs there is no region that resembles a wake region which was present in the high Reynolds number boundary layers upstream of the nozzle inlets (see Figures 4.19 and 4.25 for examples of inlet boundary layers for the LU48 and LU60 nozzles in Clauser plot form). This is particularly evident in the LU60(P), NPR = 1.50 case where the velocity is still gradually rising beyond the end of the logarithmic region. The study of laminar-exit boundary layers in Warnack and Fernholz [91] showed that once the favourable pressure gradient was relaxed the near-wall region quickly recovered from the acceleration, with the mean velocity profiles collapsing on to the standard logarithmic law of the wall. However, their results also showed that the outer region of the boundary layer, where velocity gradients are small and the
4.4 Influence of Addition of a Parallel-Walled Nozzle Exit Extension

Effects of viscosity are negligible, was very slow to respond to the removal of the favourable pressure gradient. The absence of the wake region suggests that, like the work of Warnack and Fernholz, in the present investigation the length of the relaxation region is insufficient for the outer edge to fully recover.

The effects of the parallel extension on the axial turbulence intensity are shown in Figures 4.56 and 4.57. Both nozzles exhibit similar changes (decreased near-wall peak values and a fuller, more zero-pressure-gradient-like profile) compared to the nozzles without the parallel-walled extensions. Peak values in the near-wall region have decreased from \( \approx 0.18U_b \) at \( \approx 0.12\delta_\ast^\eta \) (NPR = 1.50) and \( \approx 0.25U_b \) (NPR = 1.88) to \( \approx 0.15U_b \) at \( \approx 0.60\delta_\ast^\eta \) for both nozzles and NPRs, which represents a factor of 2 decrease for the NPR = 1.88, LU60(P) case and corresponds to decreased wall shear (inferred from Clauser plots). Peak turbulence values in the underexpanded cases (not illustrated in the figures due to the presence of the expansion fan), non-dimensionalised by \( U_b \) were slightly higher at 0.17\( U_b \) and 0.18\( U_b \) for the LU48(P) and LU60(P), but still considerably lower compared to the contraction-only nozzle results. Figures 4.58 and 4.59 show the effects of adding the parallel extension to profiles of \( \overline{u'v'} \). The results for both nozzles show recovery of Reynolds shear stress below \( y/\delta_\ast^\eta \leq 5 \) and peak values similar for both nozzles at both subsonic NPRs, which is in contrast to the results of the contraction-only nozzles where magnitudes of the near-wall peaks varied considerably with NPR and large differences between the two nozzles were present.

The results of adding the parallel extension have demonstrated the rapid recovery of the near-wall region of the highly accelerated boundary layer within the nozzle after the removal of the favourable pressure gradient. These results can be contrasted with those of Lepikovsky et al. [5] whose nozzle was contoured such that the last section also became a constant area duct. Although the acceleration within the contracting region of the Lepikovsky nozzle was sufficient that relaminarisation would probably have occurred and been sustained over a large proportion of the nozzle's length (particularly at low NPRs at high total temperature) the current results and the data presented by Warnack and Fernholz [74] suggest that, even if the boundary layer had relaminarised within the contraction region, the boundary layer could recover and become turbulent again within the constant area section of the nozzle. The laminar exit boundary layers encountered at low NPRs suggests that the boundary layer at exit was naturally laminar with laminar-turbulent transition within the nozzle being suppressed by the effects of the favourable pressure gradient. It is likely that once the Reynolds number of the nozzle is high enough transition occurs within the constant area section close to the nozzle exit. This agrees with the observation of Lepikovsky et al. that transition (based on decreasing nozzle exit boundary layer shape factor) occurred at the same jet Reynolds number regardless of NPR or total temperature.
4.5 Influence of Nozzle Exit Boundary Layers on Jet Plume Development

The previous sections have demonstrated the influence that the nozzle geometry can exert on the nozzle exit boundary layer. Despite a two orders of magnitude reduction in the momentum thickness Reynolds number and increases in the shape factor toward more laminar-like values, boundary layers at the nozzle exit were still turbulent (albeit non-equilibrium) with much thicker, higher Reynolds number boundary layers which approached the fully-turbulent state (although still not in equilibrium) achieved after the addition of the parallel-walled extension pieces. Although several comparisons between the development of initial regions of axisymmetric jets originating from laminar and turbulent nozzle exit boundary layers exist in the current literature (as discussed in Section 1.3.3) these earlier studies have been conducted at extreme conditions (i.e originating from fully-developed pipe flows). Much less information exists on the development of jet plumes originating from nozzles with fully turbulent boundary layers but with widely varying momentum thickness Reynolds numbers which are more typical of small-scale laboratory tests. For this reason, a study of the development of the jet plume originating from a contraction-only geometry and a nozzle with a parallel-walled extension piece was performed using the LDA technique. Since similar trends in the behaviour of the nozzle exit boundary layers were observed for both nozzle scales and across a wide range of NPRs, it was decided to limited the number of test conditions to three and use only one nozzle size. Tests were conducted at the same NPRs as investigated for the study of parallel-walled extensions (NPR = 1.50, 1.88 and 2.40). The LU48 and LU48(P) nozzles were selected on the grounds of their smaller mass flow rates, as it was desirable to maximize the duration of each test in order to maximize the number of samples and measurement locations. The two subsonic conditions are compared and contrasted first to investigate the influence of NPR. The shock-containing, underexpanded case (NPR = 2.40) is considered separately.

4.5.1 Near-field Development of Jet Plumes - Subsonic NPRs

Figure 4.60 shows a comparison of the axial development of the mean and RMS of the axial velocity, non-dimensionalised with respect to the axial velocity in the potential core (used to allow comparison between data gathered on different days and slightly different operating conditions), for a pressure ratio of 1.50 for both nozzle geometries along the jet centreline and along the lip line (where turbulence levels are expected to be at their maximum); this provides valuable information about the transformation of the nozzle exit boundary layer into the jet free shear layer [60]. The first measurement point was located 4mm, \((x/D = 0.0833)\)
downstream of the nozzle exit to ensure an unobstructed beam path. The profiles show that, despite large variations in the shape and thickness of the nozzle exit boundary layer from which the axisymmetric free shear layer is developing, the centreline development of the mean velocity is virtually identical for the two geometries in terms of the length of the potential core (taken as the point at which $U = 0.99U_{pc}$ which corresponds to $x/D = 6.2$) and the decay rate of the centreline velocity. The only obvious variation between the two geometries and corresponding exit conditions occurs in the first jet diameter. Velocities at the exit of the contraction-only nozzle are initially 10% lower than the plateau in the centreline velocity and rapidly increase over the first $0.75D$. The increase in the velocity results from the presence of a vena contracta whereby the cross section of the jet plume decreases and the fluid continues to accelerate in the centreline region beyond the end of the nozzle. Beyond the end of the vena contracta the velocity of the fluid within the potential core is the same for both contraction-only and parallel-walled extension nozzles. The vena contracta is an inviscid effect resulting from inward radial velocities generated within the nozzle and has been removed through the addition of the parallel extension. The vena contracta plays a significant role in the development of the near field region of the jet as will be shown below.

The variation of the RMS of the axial velocity along the centreline does not show any influence of the vena contracta, despite the additional inviscid strain rate $\frac{\partial U}{\partial z}$ that is produced as the flow is accelerated. The profiles collapse well, throughout the potential core region and the developed region beyond its end. Some variation can be seen beyond $x/D \approx 8$ and is attributed to experimental errors. Noteworthy also is the behaviour of the RMS profiles in the region $1.5 < x/D < 6$, which is within the potential core. $U_{RMS}$ can be seen to increase linearly in this region at a smaller gradient than is observed when the shear layer closes at the end of the potential core where turbulence levels rise more rapidly. This behaviour (increasing turbulence level) is not expected within the potential core where both axial and radial velocity gradients are negligible suggesting that no turbulence production is possible. It is not believed that the observed results are the influence of fluctuations in the supply pressure since, if this was the case, the increase in the centreline turbulence would be expected to start from the nozzle exit and not some distance downstream of it. Similar variations appeared in the LDA measurements of high subsonic axisymmetric jets ($M_s = 0.75, 0.9$) by Power et al. [154]. Their measurements of the energy spectrum of the axial RMS velocity on the centreline within the potential core revealed a low frequency, narrow band peak which they suggested was the result of local irrotational unsteadiness induced by the streamwise passage of large turbulent structures in the initial regions of the shear layer. Unlike the centreline profiles, profiles along the lipline do show significant variation between the two nozzle geometries, which is not unexpected, given the evidence presented above of their very different nozzle exit boundary layer characteristics. The lipline
profile of the axial mean velocity for the \(LU48(P)\) nozzle shows a small decrease in magnitude beyond the end of the nozzle from \(0.64U_{pc}\) to \(0.62U_{pc}\), followed by an increase to \(0.69U_{pc}\) as the jet spreads and the centre of the shear layer moves outwards, with finally a linear decrease beyond \(x/D \approx 2\) due to entrainment of ambient fluid into the jet. The profiles of the \(LU48\) nozzle shows much lower velocities at the first measurement location (\(0.2U_{pc}\)) peaking at \(0.62U_{pc}\) at \(x/D \approx 2.5\). This behaviour can be attributed to the more laminar-like exit boundary layer and the presence of the vena contracta. The acceleration of the flow beyond the end of the nozzle and resulting reduction in the cross section of the jet will draw the developing shear layer inwards toward the centreline. A linear decrease is still eventually observed, but the rate of decrease is lower than seen in the case of the \(LU48(P)\) nozzle for the whole \(10D\) distance measured.

Lipline profiles of axial RMS velocity also show significant variation. The \(LU48\) nozzle reaches a peak of \(0.165U_{pc}\) compared to \(0.154U_{pc}\) for the \(LU48(P)\), a difference of 7%. This difference is short-lived and by \(x/D \approx 6\), the two results collapse. Bradshaw [48] noted that the turbulence intensity variation along the lip line varied depending on the state of the boundary layer. If the boundary layer was laminar or transitional the turbulence levels increased to a local maximum close to the lip. If the boundary layer was fully turbulent, turbulence levels were seen to grow gradually with axial distance, reaching a peak a few nozzle exit diameters downstream. Similar behaviour was also seen in the work of Hill et al. [60]. The profile from the contraction only nozzle shows a rapid increase to its peak at \(x/D \approx 0.75\) followed by a gradual decrease. For the nozzle with the parallel extension, whose exit boundary layer has been demonstrated to be turbulent, values are seen to rise rapidly over the same distance and continue to increase very slowly up to around \(x/D \approx 6\). The growth in turbulence intensity for this case is certainly not as pronounced as the results of Hill[60] or Bradshaw[48], whose results show clear peaks several diameters downstream. The results of Hill show turbulence levels continuing to grow beyond \(x/D = 6\). Based on the observations of Hill [60] and Brawshaw[48], it is argued that the \(LU48\) results demonstrate transitional behaviour with a local maximum close to the nozzle exit (this is consistent with the boundary layer measurements which showed transitional shape factors, Reynolds numbers and profiles). The results of \(LU48(P)\) nozzle, despite having an exit boundary layer with a developed turbulent near-wall region, also appears to demonstrate laminar or transitional behaviour along the lip line. This suggests that the outer region of the boundary layer, which Warnack [91] showed to recover very slowly after the acceleration was relaxed, is important in the development of the jet free shear layer.

Data for \(NPR = 1.88\) are given in Figure 4.61 and essentially show the same behaviour discussed above for the lower NPR. There are, however, two noticeable variations in the results. First, the length of the potential core has increased from \(6.2D\) to \(6.8D\). An increase
4.5 Influence of Nozzle Exit Boundary Layers on Jet Plume Development

is expected as the work of Lau [106] showed that the length of the potential core for a compressible axisymmetric jet increases as a function of jet Mach number but the current values are much larger than those predicted by the correlation of Lau[106] which predict lengths of $4.9D \ (NPR = 1.50)$ and $5.3D \ (NPR = 1.88)$. Comparing the rate of change of potential core length growth with jet Mach number the present results increase at a rate 1.5 times greater than predicted by Lau[106]. The second difference is the effect of the vena contracta. At the higher $NPR = 1.5$, with a value of $0.83U_{pe}$ compared to $0.907U_{pe}$; the increase in the axial velocity beyond nozzle exit is also more rapid so that the axial extent of the vena contracta has been halved from $0.66D$ to $0.33D$.

Axial profiles along two lines have shown that although there are only small differences along the jet centrelines there are significant differences along the lipline. To provide better insight into the shear layer development radial profiles are better suited. Measurements took advantage of the axisymmetry of the jet plume by measuring on one radial side only. This allowed measurements to be taken at approximately 70 radial locations allowing good spatial resolution of the developing shear layer, particularly in the region close to the nozzle exit where the shear layer was very thin. The precise location of the centreline was established through a series of traverses (both horizontally and vertically) across the jet close to the nozzle exit. Measurements made at $x/D = 10$ to identify the location of the centreline showed an angular misalignment of the jet axis relative the the LDA measurement axis corresponding to less than $0.2^\circ$ which was deemed insignificant.

Figures 4.62-4.64 show the development of the radial profiles of axial mean and RMS velocity and Reynolds shear stress at 12 axial locations up to $x/D = 10$ for LU48 and LU48(P) nozzles at $NPR = 1.50$. At the first two measurement locations the effects of the vena contracta can be seen in the plume originating from the contraction only (LU48) nozzle (Figure 4.62). At $x/D = 0.25$ the centreline axial velocity is only at $0.96U_{pe}$, but by $x/D = 0.50$ a top-hat profile has developed although the maximum velocity is still below that in the region of the potential core beyond the vena contracta. There is no evidence of a vena contracta in the plume originating from the LU48(P) nozzle. Focusing on the shear layer region it is seen that the profiles from the LU48 geometry constantly lie below those from the LU48(P) nozzle, denoting a narrower jet plume. This behaviour is still observed as far as $x/D = 10$, despite the agreement on the centreline velocity shown in Figure 4.60. Figure 4.63 shows the axial development of the non-dimensional radial $u_{RMS}$ profiles. These give a good insight into the differences between the two jet plumes. Profiles at $x/D = 0.25$ and $x/D = 0.50$ indicate noticeable variations. At $x/D = 0.25$ a peak value of $0.153U_{pe}$ is observed for LU48 compared to the smaller peak of $0.144U_{pe}$ for LU48(P). The LU48(P) profile is also wider and shifted outboard. As the downstream axial distance increases, the
4.5 Influence of Nozzle Exit Boundary Layers on Jet Plume Development

difference in the peak magnitudes also decreases; by $x/D = 1.0$ peak levels ($0.17U_{pc}$) have become indistinguishable. Beyond $x/D = 1.00$ peak levels decrease gradually. The peak location in the $LU48$ data lie closer to the centreline compared to the $LU48(P)$ results and at all stations down to $x/D = 10$ the $LU48(P)$ results are shifted slightly outboard of the $LU48$ data, although by $x/D = 10$ the shift is no longer significant. Similar behaviour as observed in the axial RMS profiles is visual in the radial profiles of the shear stress $\bar{u}'\bar{v}'$, shown in Figure 4.64. Both jet plumes show similar behaviour, with peak levels increasing over the first 2 diameters, followed by a gradual decrease.

Figures 4.65 and 4.66 show a comparison of three key lines defining the initial stage of spreading of the jet shear layer: the edge of the potential core (denoted by $U = 0.95U_{pc}$ ($U = 0.95U_{pc}$ has been chosen over $U = 0.99U_{pc}$ as this definition can not be applied in the region of the vena contracta where core velocities are below $U = 0.99U_{pc}$)), the outer edge of the jet plume (denoted by $U = 0.05U_{cl}$) as well as the jet half-radius (denoted by $U = 0.50U_{cl}$) for $NPR = 1.50$ and 1.88, respectively. Figure 4.65 illustrates the offset of the shear layer toward the centreline for the contraction-only nozzle compared to the parallel extension nozzle. Although the shear layer is thinner for the contraction nozzle, a result of the much thinner nozzle exit boundary layers, the inviscid displacement is attributed to the effect of the vena contracta associated with nozzle convergence. Comparing these results with those presented in Figure 4.66 it can be seen that the displacement of the shear layer is less for the higher NPR. As shown above, the influence of the vena contracta, both in terms of its extent and the variation between nozzle exit velocities, is considerably smaller than observed at the lower NPR.

The thickness and growth of the jet free shear layer in the initial region is now considered. Fondse et al. [54] and Hussain and Zedan [53] use the growth of the shear layer thickness ($\frac{dB}{dx/D}$) to investigate the influence of initial conditions on the development of the initial regions of the axisymmetric jet. The shear layer thickness is defined as:

$$B = r_{0.05} - r_{0.95}$$ (4.8)

where $r_{0.05}$ and $r_{0.95}$ denote the point at which the axial velocity has decreased to 0.05 and 0.95 times the centreline velocity at the given axial location, respectively.

Figure 4.67 shows the growth of the shear layer in the first 6 nozzle exit diameters of the plume for both nozzle sizes and for $NPR = 1.50$ and 1.88. For both NPRs the shear layer growth rate based on a linear fit of the shear layer thicknesses is slightly greater for the nozzle with parallel extension compared to the contraction-only nozzle, with growth rates of 0.1602 ($LU48(P)$) and 0.1543 ($LU48$) for $NPR = 1.50$ (a 3.8% variation) and 0.1627 ($LU48(P)$) and 0.1599 ($LU48$) at $NPR = 1.88$ (a 1.8% variation). The greater spreading
4.5 Influence of Nozzle Exit Boundary Layers on Jet Plume Development

Influence of Nozzle Exit Boundary Layers on Jet Plume Development

Rates for the shear layers developing from nozzle exit boundary layers is in disagreement with the study of the influence of the initial momentum thickness on the development of the shear layer by Hussain and Zedan. They argued that the small changes in the growth rate were independent of the initial momentum thickness and Reynolds number (which varied by a factor of 4, with initial nozzle exit boundary layer momentum thickness Reynolds numbers in the range $184 < Re_i < 284$). Variations in spreading rate were attributed to small changes in the initial conditions. Although there are clear differences between the growth of the shear layers for the two nozzles, the influence of the nozzle exit boundary layer appears to be small as an order of magnitude increase in $\theta_i$ and $Re_i$ has only resulted in an increase of a few percent.

4.5.2 Near-field Development of Jet Plumes - Supersonic (Under-expanded) NPRs

Once the NPR has been raised above the critical value the nozzle will choke and the jet plume enters the shock-containing underexpanded regime which was described in Chapter 1.

Figure 4.68 shows the variation of mean and RMS axial velocities along the centreline and lipline. Values have in this case been made non-dimensional with respect to the ideal fully expanded velocity (for $NPR = 2.4$, $T=$ ambient (284.6K) and $\gamma = 1.4$ this corresponds to $354 m/s$). The presence of the shock cells is illustrated along the centreline profile of axial velocity which shows repeated increases as the flow is accelerated through the expansion fans followed by a sharp decrease as the flow is decelerated by the oblique shock waves. For both geometries studied there are 10 shock cells based on the number of measured velocity peaks.

It is immediately apparent that there are significant differences in the near-field behaviour of the two jet plumes, in stark contrast to both subsonic cases which showed very close agreement beyond the end of the vena contracta. Firstly the non-dimensional velocity at the nozzle exit of the LU48 nozzle is considerably lower for the LU48(P) nozzle; $0.76U_{expanded}$ compared to $0.88U_{expanded}$. This indicates a greater degree of underexpansion toward the centreline in the plume originating from the LU48 nozzle, caused by the vena contracta. More significant is the apparent phase difference in the shock cells. The first shock cell is longer for the LU48 nozzle, with a length of $0.84D$ compared to $0.74D$. The differing shock cell length is limited to the first shock cell, with the lengths of the next 2 shock cells for both cases equal to $0.74D$ as deduced from the Schlieren image in Figure 4.70 ($\approx 0.75D$ deduced from the measured axial velocity profiles). The decrease in centreline velocity that occurs beyond the end of the potential core (taken as the point at which centreline RMS values begin to grow rapidly) can be seen to occur at a greater axial distance in the LU48 results. A similar shift in the end of the potential core is also demonstrated in the centreline...
4.5 Influence of Nozzle Exit Boundary Layers on Jet Plume Development

The variation of the RMS of the axial velocity which shows a similar delay in the location of the rapid rise in turbulence levels at the end of the potential core as the shear layer closes \((x/D \approx 5.7\) for \(LU48(P)\) and \(x/D \approx 6.0\) for \(LU48\)). The centreline RMS profiles show large fluctuations in magnitude within the potential core region corresponding to the locations of oblique shock waves and expansion fans. The same fluctuations do not occur in the profiles of \(u_{RMS}\) (Figure 4.69) and, therefore, the peaks are most likely an artifact introduced through the effects of seeding particle velocity lag and large axial velocity gradients across the LDA measurement volume in the region of the shock waves and expansion fans.

Turning attention to the variation along the lipline, differences in mean velocity are only apparent over the first two diameters downstream where the shear layer is thin and is moving toward and away from the centreline as the jet contracts and expands (shown in Figure 4.70, which will be discussed in more detail below). Beyond \(x/D \approx 2\) where the lipline flow is subsonic the profiles show close agreement. Despite the agreement between velocities, RMS values are consistently around 20% higher for the contraction-only nozzle.

Figure 4.70 shows a comparison of two colour Schlieren images of the first 3 nozzle exit diameters of the jet plume for the \(LU48(P)\) (left side) and \(LU48\) (right side). Both images are to the same scale. Regions of expansion (negative axial density gradients) are indicated by orange and regions of compression (positive axial density gradients) are indicated by blue. All other regions appear as green. These images highlight the difference in the shock cells seen above in the variation in the mean velocities along the centreline. The greater length of the first shock cell in the \(LU48\) case is immediately apparent, as is a larger region of expansion of the flow immediately downstream of the nozzle exit, particularly near the centreline, as denoted by the larger orange region which is also more convex in appearance. The variation of the first shock cell is certainly a consequence of the vena contracta. As also seen in the subsonic cases, the fluid close to the centreline has not undergone the same degree of acceleration in the contraction-only nozzle compared to the parallel-extension geometry (illustrated by the difference in centreline velocities at the first measurement point in Figure 4.68). This means that the static pressure at the exit centreline of the contraction-only nozzle is much higher and, therefore, a greater degree of expansion is required in order to reduce the static pressure to ambient. As the amount of expansion increases, the angle subtended by the expansion fan relative to the centreline increases and the length of the shock cell also increases. The nature of the oblique shock wave formed by the coalescence of the reflected Mach waves that form the expansion fan in the first shock cell are also different. The subtended angle for the \(LU48\) nozzle is greater than for \(LU48(P)\) nozzle indicating greater recompression in the \(LU48\) case.

The effects of the vena contracta are also very apparent in the radial profiles of axial
velocity presented in Figure 4.71. Large variations between the two jets can be seen within the supersonic region depending on whether the measurement locations correspond to the location of an expansion, recompression or oblique shock region. Significant variations at the centreline are still present up to \( x/D \approx 8 \). Despite large variations in the supersonic region, the velocity distributions within the shear layer itself follow similar trends. For axial locations beyond \( x/D = 2 \), the shear layer profiles collapse. Some of the apparent displacement of the shear layer is attributed to the local contraction and expansion of the jet plume. Figures 4.72 and 4.73 present radial profiles of the axial RMS velocity and \( u'v' \) correlation. Like the mean velocity profiles, good agreement between the two jets is seen on the outer edge of the shear layer beyond \( x/D = 2 \) for both \( u_{RMS} \) and \( u'v' \). However, peak values are different at all of the measurement locations with larger peaks values occurring in the profiles of the LU48 jet. There is one exception to this observation, at \( x/D = 0.50 \) (which is within the first shock cell) the peak values for the LU48(P) are greater.

The results for the underexpanded jet plumes suggest that nozzle exit boundary layers play an indirect role in the near field development of an underexpanded jet plume as large variations in the characteristics of exit boundary layer between cases were not reflected in the mean velocity and turbulence statistics within the shear layer. However, the results do show that geometric effects do significantly influence the development of the jet plume. The formation of a vena contracta due to contraction of the nozzle geometry significantly influenced the size and strength of the first shock cell and the location of subsequent shock cells due to a greater degree of underexpansion at the nozzle exit centreline. The results suggest that nozzle geometry (in particular its convergence) is important in controlling inviscid phenomena in underexpanded jet plume development.

### 4.6 Closure

In this chapter nozzle jet flow measurements using a combination of intrusive and non-intrusive measurement techniques have been presented, including measurements of the nozzle inlet approach flow. Inlet measurements showed that high Reynolds number, developed turbulent boundary layers were achieved for all nozzle scales across the whole range of operating conditions.

Nozzle exit measurements using two independent measuring techniques have been presented. The data have shown that the acceleration through the nozzle causes drastic changes to the thickness and shape of the boundary layer which becomes more laminar-like due to the effects of the static pressure gradient within the nozzle. The changes in the boundary layer thickness and shape were also shown to be greater with increasing NPR. The effects
of the addition of a parallel extension at nozzle exit were also presented. This addition was shown to return the boundary layer to a high Reynolds number, turbulent state and significantly increase its thickness in a short distance. The difference between results for two nozzle geometries show that the recovery is influenced by nozzle geometry particularly the factor of 2 difference in $Re_{eq}$ between the two different geometries despite similar exit boundary layer characteristics for the corresponding contraction-only geometries. The data may now be used to provide inlet conditions and benchmark data for a CFD study that can explain the reasons for the observed changes in the state and thickness of the measured boundary layers.

The results have further brought into question the nozzle exit boundary layer data of Lepicovsky et al. [5], since the present data have clearly demonstrated that the acceleration within laboratory-scale nozzles is capable of producing low Reynolds number laminar-like exit boundary layers despite fully turbulent high Reynolds number inlet boundary layers.

Measurements of the initial region of jet plumes originating from nozzles with widely different boundary layers have shown that despite order of magnitude differences in the kinematic momentum thickness Reynolds number of the nozzle exit boundary layers, the effects on the development of high speed, high Reynolds number jet plumes were isolated to the free shear layer in the first few nozzle exit diameters of the plume. This suggests that variations in turbulent exit boundary layers may be considered as a secondary influence on jet plume development. This is in agreement with the work of Hill et al. [60] who also showed jets with turbulent exit boundary layers were unaffected by variations in the exit boundary layer. However, the effects of the boundary layers on the first jet diameter or so of the jet shear layer will be of interest to those studying jet plume development and mixing enhancement in the region very close to the nozzle exit, for example novel nozzle configurations such as tabs or serrations. The plume measurements also showed that the additional, inviscid geometric effects (the presence of a vena contracta) do significantly affect the development of the initial region, particularly when operating at supercritical (underexpanded) pressure ratios where variations in exit centreline static pressures caused by the vena contracta were seen to significantly alter the development of the shock cells within the supersonic core region which is significant when considering flows where the shock structures are important, for example, IR observability [15] and shock noise studies [11].
4.7 Figures
Figure 4.1: Variation of LU48 Nozzle Inlet Boundary Layer Parameters with NPR
Figure 4.2: Variation of LU60 Nozzle Inlet Boundary Layer Parameters with NPR

(a) Kinematic Momentum Thickness Reynolds Number

(b) Kinematic Momentum Thickness

(c) Kinematic Shape Factor

(d) Kinematic Skin Friction
Figure 4.3: Influence of Compressibility on Shape Factor (LU60)

Figure 4.4: Influence of Compressibility on Momentum Thickness Reynolds Number (LU60)
4.7 Figures

Figure 4.5: Correlation of Local Skin Friction Coefficient with Kinematic Momentum Thickness Reynolds Number (LU48, LU60)
Figure 4.6: LU48 mean velocity NPR 1.3-1.8 (Inner Wall Scaling)

Figure 4.7: LU48 mean velocity NPR 1.8-2.4 (Inner Wall Scaling)
Figure 4.8: LU60 mean velocity NPR 1.3-1.8 (Inner Wall Scaling)

Figure 4.9: LU60 mean velocity NPR 1.8-2.4 (Inner Wall Scaling)
Figure 4.10: LU48 mean velocity NPR 1.3-1.8 (Outer Wall Scaling)

Figure 4.11: LU48 mean velocity NPR 1.8-2.4 (Outer Wall Scaling)
4.7 Figures

Figure 4.12: LU60 mean velocity NPR 1.3-1.8 (Outer Wall Scaling)

Figure 4.13: LU60 mean velocity NPR 1.8-2.4 (Outer Wall Scaling)
Figure 4.14: LU48 Nozzle Exit Boundary Layer Kinematic Momentum Thickness Reynolds Number Variation (Probe Data)

Figure 4.15: LU48 Nozzle Exit Boundary Layer Kinematic Momentum Thickness Variation (Probe Data)
Figure 4.16: LU48 Nozzle Exit Boundary Layer Kinematic Shape Factor Variation (Probe Data)

Figure 4.17: LU48 Nozzle Exit Boundary Layer Profile Variation (NPR 1.3-1.6) (Probe Data)
Figure 4.18: LU48 Nozzle Exit Boundary Layer Profile Variation (NPR 1.65-1.9) (Probe Data)

Figure 4.19: LU48 Nozzle Exit Boundary Layer Clauser Plots (Probe Data)
Figure 4.20: LU60 Nozzle Exit Boundary Layer Kinematic Momentum Thickness Reynolds Number Variation (Probe Data)

Figure 4.21: LU60 Nozzle Exit Boundary Layer Kinematic Momentum Thickness Variation (Probe Data)
Figure 4.22: LU60 Nozzle Exit Boundary Layer Kinematic Shape Factor Variation (Probe Data)

Figure 4.23: LU60 Nozzle Exit Boundary Layer Profile Variation (NPR 1.3-1.6) (Probe Data)
4.7 Figures

Figure 4.24: LU60 Nozzle Exit Boundary Layer Profile Variation (NPR 1.65-1.9) (Probe Data)

Figure 4.25: LU60 Nozzle Exit Boundary Layer Clauser Plots (Probe Data)
Figure 4.26: Influence of LDA Measurement Volume on Mean Axial Velocity

Figure 4.27: Influence of LDA Measurement Volume on Clauser Plot Mean Axial Velocity
Figure 4.28: Influence of LDA Measurement Volume on the RMS of Mean Axial Velocity
Figure 4.29: Influence of Weighting on Mean Velocity (LU60, NPR1.25)

Figure 4.30: Influence of Weighting on RMS Axial Velocity (LU60, NPR1.25)
Figure 4.31: Influence of Weighting on Reynolds Shear Stress (LU60, NPR1.25)

Figure 4.32: Influence of Weighting on Kinematic Shape Factor (LU60)
Figure 4.33: Influence of Weighting on Kinematic Momentum Thickness Reynolds Number (LU60)

Figure 4.34: LU48 LDA Nozzle Exit Boundary Layer Kinematic Momentum Thickness Reynolds Number Variation
Figure 4.35: LU48 LDA Nozzle Exit Boundary Layer Kinematic Momentum Thickness Variation

Figure 4.36: LU48 LDA Nozzle Exit Boundary Layer Kinematic Shape Factor Variation
Figure 4.37: LU48 LDA Nozzle Exit Boundary Layer Profile Variation (NPR 1.3-1.75)
4.7 Figures

Figure 4.38: LU48 LDA Nozzle Exit Boundary Layer Profile Variation (NPR 1.8-2.4)

Figure 4.39: Comparison of Nozzle Inlet and Nozzle Exit Boundary Layer profiles (LU48)
Figure 4.40: LU48 LDA Nozzle Exit Boundary Layer RMS Axial Velocity Profile Variation (NPR 1.3-1.75)

Figure 4.41: LU48 LDA Nozzle Exit Boundary Layer Reynolds RMS Axial Velocity Profile Variation (NPR 1.8-2.40)
Figure 4.42: Variation of Peak RMS with NPR (LU48 and LU60)
Figure 4.43: LU60 LDA Nozzle Exit Boundary RMS Axial Velocity Profile Variation (NPR 1.3-1.75)

Figure 4.44: LU60 LDA Nozzle Exit Boundary RMS Axial Velocity Profile Variation (NPR 1.8-2.4)
Figure 4.45: LU48 LDA Nozzle Exit Boundary Layer Reynolds Shear Stress ($u'v'$) Profile Variation (NPR 1.3-1.75)

Figure 4.46: LU48 LDA Nozzle Exit Boundary Layer Reynolds Shear Stress ($u'v'$) Profile Variation (NPR 1.8-2.40)
Figure 4.47: Variation of Peak Reynolds Shear Stress ($u'v'$) with NPR (LU48 and LU60)
Figure 4.48: LU60 LDA Nozzle Exit Boundary Reynolds Shear Stress ($u'v'$) Profile Variation (NPR 1.3-1.75)

Figure 4.49: LU60 LDA Nozzle Exit Boundary Reynolds Shear Stress ($u'v'$) Profile Variation (NPR 1.8-2.4)
Figure 4.50: Location of LDA Traverse Line Relative to Nozzle Lip Expansion Fan (LU60, \(NPR = 2.40\))
Figure 4.51: Influence of Prandtl-Meyer Expansion Fan on Measured Boundary Layer Velocity Profile (LU60(P))

Figure 4.52: LU48(P) Nozzle Exit Boundary Layer Profiles
Figure 4.53: LU60(P) Nozzle Exit Boundary Layer Profiles

Figure 4.54: LU48(P) Clauser Plots
Figure 4.55: LU60(P) Clauser Plots

Figure 4.56: LU48(P) Nozzle Exit Boundary Layer RMS Axial Velocity Profiles
Figure 4.57: LU60(P) Nozzle Exit Boundary Layer RMS Axial Velocity Profiles

Figure 4.58: LU48(P) Nozzle Exit Boundary Layer Reynolds Shear Stress ($u'v'$) Profiles
Figure 4.59: LU60(P) Nozzle Exit Boundary Layer Reynolds Shear Stress ($u'v'$) Profiles
Figure 4.60: Comparison of Axial Variation of Mean and RMS Axial Velocity Profiles (NPR = 1.50)
Figure 4.61: Comparison of Axial Variation of Mean and RMS Axial Velocity Profiles (NPR = 1.88)
Figure 4.62: Profiles of Axial Velocity (NPR = 1.50)
Figure 4.63: Profiles of Axial RMS Velocity (NPR = 1.50)
Figure 4.64: Profiles of Shear Stress $u'v'(NPR = 1.50)$
Figure 4.65: Jet Plume Comparison (NPR=1.50)
Figure 4.66: Jet Plume Comparison (NPR=1.88)

Figure 4.67: Growth of Mixing Layer
Figure 4.68: Comparison of Axial Variation of Mean and RMS Axial Velocity Profiles (NPR = 2.40)
Figure 4.69: Axial Variation of $V_{rms}$ (NPR = 2.40)
Figure 4.70: Schlieren Image of Underexpanded Jet Plumes
Figure 4.71: Profiles of Axial Velocity (NPR = 2.40)
Figure 4.72: Profiles of Axial RMS Velocity (NPR = 2.40)
Figure 4.73: Profiles of Shear Stress $\overline{u'v'}$ (NPR = 2.40)
Chapter 5

Numerical Prediction of Nozzle Boundary Layer Development

5.1 Introduction

The previous chapter has highlighted the large changes that occur to the nozzle wall boundary layer as it negotiates the acceleration through the contraction region of the nozzle, with reductions of two orders of magnitude in the kinematic momentum thickness Reynolds number (despite large increases in velocity), and shifts in the mean velocity profile from a turbulent shape toward a more laminar-like one, as illustrated by large increases in the kinematic shape factor. The addition of a parallel-walled extension piece at the end of the contraction was shown to return the boundary layer rapidly to a developed turbulent shape and a high Reynolds number. The present chapter aims to explain the changes in the boundary layer observed experimentally using the numerical methods described in Chapter 3.

Since similar trends were observed for both nozzle scales investigated and across a wide range of NPRs, the simulations were limited to three representative operating conditions. The subsonic cases are $NPR = 1.50$ and $NPR = 1.88$. Beyond the critical NPR the Mach number and, hence, velocity variation within the nozzle would be small with further NPR increases (neglecting small changes in the blockage effect of the boundary layer). The inlet measurements in the previous chapter had shown that beyond the point of choking the kinematic momentum Reynolds number varied slowly with NPR, mainly as a result of increased density, as measured velocities at the centreline remained near-constant for supercritical NPRs. For this reason a single underexpanded condition was chosen, $NPR = 2.4$, which corresponded to the highest NPR investigated experimentally. A single nozzle
scale was chosen, namely the contracting LU60 nozzle and the LU60(P) nozzle with its parallel-walled extension, since this had shown the greatest changes in the boundary layer between inlet and exit in the experimental study reported in Chapter 4.

The remainder of the present chapter describes background details of the numerical predictions of the development of the boundary layer through the nozzle using the Launder-Sharma low Reynolds number turbulence model as described and validated in Chapter 3. The following sections therefore describe: the approach used for the geometry definition and simplification of the solution domains for the two nozzles simulated (Section 5.2), the domain boundary types and generation of domain inlet conditions used (Section 5.3). The simulations were mainly restricted to the flow within the nozzle only so that the exit plane of the solution domain was located at the nozzle exit plane. The influence of this choice on the predicted flowfield was addressed by examination of predictions for the LU60 (contraction only) nozzle against simulations which included a short distance of the emerging jet plume (Section 5.4). Finally, in Section 5.5 the mechanisms responsible for the high momentum thickness Reynolds number and turbulent shape factor achieved through the addition of a parallel-walled nozzle extension were investigated.

5.2 Domain Geometries and Meshing

The domain geometries used for the LU60 and LU60(P) nozzles are shown in Figures 5.1 and 5.2. For all of the nozzle simulations performed, a 15° sector was used to take advantage of the axisymmetry of the geometries. The solution domain started 0.5 nozzle delivery duct diameters upstream of the point at which the nozzles were attached to the primary delivery duct. This location corresponded to the location of the experimental inlet boundary layer measurements that would be used for the nozzle inlet conditions in the simulations. The same approach to avoiding poor convergence in the centreline region (caused by very small element lengths in the azimuthal direction at the centreline when using a polar mesh) as used in the validation of the low Reynolds number turbulence model in Section 3.8 was adopted for all simulations. This included use of a centreline ‘root’ surface, with a radius of 1.5mm, which corresponds to 2.5% of the nozzle exit diameter and a negligible blockage of 0.0625% of the nozzle exit area; it was, therefore, not expected to influence significantly the flow within the nozzle. For numerical stability reasons it was found that simulated geometries required a small modification from those tested experimentally. Previous simulations of internal nozzle flows by Trumper [62] had shown odd-even pressure-velocity decoupling within the nozzle. This decoupling resulted from the use of a sharp corner (i.e. a slope discontinuity) in the nozzle wall at the junction between the parallel delivery duct and the contracting nozzle. The slope discontinuity involves very large local pressure gradients which then induced 2Δx
oscillations in the velocity field. These could have been suppressed by increasing the Rhie and Chow smoothing, but it was decided instead to modify the geometry slightly. A fillet of 15 mm radius was introduced to remove this problem (as can been seen in the solid model representations shown in Figures 5.1 and 5.2). In the case with a parallel extension no odd-even decoupling was encountered at the joint between the contraction and constant area section so the addition of a filleted region was not required. It is worth noting that during manufacture of the nozzles it was not possible to create a sharp transition into the contraction region of the nozzle and as a result the experimental nozzles would also feature a fillet region. The study of the influence of the domain exit involved a third geometry (Figure 5.3) which included a small region of the jet plume (extending approximately 4 nozzle exit diameters in the axial direction and 3 exit diameters in the radial direction) which allowed the domain exit to be located well away from the nozzle exit.

Figures 5.4 -5.6 show two-dimensional slices through typical meshes used for the present investigations. For all simulations several versions of these meshes were explored in order to ensure results were mesh-independent. Particular attention was paid to near-wall resolution, ensuring that the first cell centre was located at a wall normal distance of $y^+ \approx 0.1$, with cell sizes increasing gradually following an exponential distribution. It was also ensured that there were at least 40 points within the region $y^+ < 30$ and approximately 100 points within the boundary layer in accordance with the findings of the radial mesh sensitivity study in Section 3.8 and the recommendations of Bardina et al. [149]. An exponential expansion of cells was also used in the axial direction moving upstream of the nozzle exit plane to allow greater mesh densities near the end of the contraction where the flow was expected to accelerate very rapidly. For simulations of the contraction-only nozzle, the final mesh comprised of a single block of $200 \times 150 \times 10$ cells in the axial, radial and azimuthal directions. For simulations of a nozzle with a parallel wall extension, a multiblock approach was used to provide better control of mesh quality in the parallel section in order to resolve the flow phenomena associated with the vena contracta. The meshing parameters used in the contracting region of the nozzle were identical to the contraction-only case. Two additional blocks were used for mesh generation of the parallel extension (see Figure 5.5). In total the mesh comprised of $265 \times 150 \times 10$ cells in the axial, radial and azimuthal directions. In addition to the internal nozzle mesh, the plume mesh (Figure 5.6) included two additional blocks within the jet plume region which increased the total mesh size to approximately 400000 cells.
5.3 Boundary Conditions

As experimental inlet data were available for the NPRs to be used in the present simulations, a similar approach to inlet condition generation as used for the relaminarising boundary layer test case in Section 3.8 was used again. A 15° segment of the nozzle delivery pipe, 2m in length, was simulated using a specified, radially-invariant total pressure (fixed by the NPR of the simulation) and low levels of $k$ and $\bar{e}$ at its inlet, and a fixed static pressure at the nozzle exit.

Inlet conditions for these precursor calculations were based on experimental operating conditions. Inlet values of $k$ and $\bar{e}$ were based on measured centreline velocities and an assumed turbulence intensity of 2% and a length scale corresponding to the diameter of the supply duct. Mirror symmetry boundary conditions were used on the centreline root and sides of the solution domain. The domain exit used a fixed static pressure boundary condition. For the two subsonic pressure ratios used, this pressure was set equal to the ambient static pressure as measured on the day in the experimental study. Beyond the critical NPR the nozzle becomes choked at exit and the exit static pressure can no longer be assumed to be equal to ambient static. For the simulations at $NPR = 2.40$ the exit static pressure was derived from the relationship:

$$p_{exit} = \frac{NPR \times p_{amb}}{\left(1 + \frac{2}{\gamma - 1} M_e^2\right)^{\frac{\gamma - 1}{\gamma}}},$$  \hspace{1cm} (5.1)

where $M_e$ is the Mach number at the nozzle exit plane which was assumed equal to unity when the nozzle choked.

After convergence, the developing boundary layer in this straight pipe segment was examined and an axial location was chosen where the predicted axial velocity profile matched the experimentally-measured one (Chapter 4). Total pressure, turbulent kinetic energy and dissipation rate profiles were then extracted from the straight pipe section at this axial location and used as inlet conditions for the main nozzle simulations. The profiles were applied to the solution domain nozzle inlet boundary using the user-specified inlet boundary condition approach. Unlike for the incompressible validation test case (which only required a straight pipe to be simulated) the compressible nozzle flow simulations under consideration here required the nozzle to be included in the geometry of these precursor simulations to ensure the correct mass flow rate, which in turn ensured the correct bulk velocity and density within the precursor simulations, since these vary greatly between nozzle scales. A comparison between experimentally-measured and computationally-derived nozzle inlet conditions is presented in Table 5.1. The values in Table 5.1 show that the experimental and computational nozzle inlet boundary layer parameters agree closely, with the kinematic momentum...
thickness Reynolds number, kinematic momentum thickness and kinematic skin friction all being within 4% of measured values. Examples of precursor simulation generated nozzle inlet profiles (U, P, k and \( i' \)) for NPR = 1.50 are shown in Figure 5.7.

The exit boundary location and conditions are important issues to be considered when studying the flow within the nozzle. Ideally, exit boundaries should be located as far away as possible from the region of interest to ensure ‘boundary condition transparent’ solutions since exit boundary conditions are often approximations (e.g., in reality the exit static pressure may not be exactly uniform at the location of the fixed static pressure boundary over the whole nozzle radius). However, in the current investigation, the use of a low Reynolds number turbulence model results in large mesh sizes and fine near-wall meshes (particularly as the bulk average jet Reynolds number is high and the boundary layer at the nozzle exit has been shown to be highly suppressed in the previous chapter). Therefore, to avoid extra computational expense any additional solution domain to move the exit boundary away from the region of interest needs to be minimised. For the nozzle flow simulations therefore the domain exit was initially located at the nozzle exit plane. This was not expected to influence the parallel extension simulations since axial variations of flow properties were expected to be reasonably small. In the contraction-only geometry cases the same may not be true due to the presence of the vena contracta as identified in the experimental jet plume measurements.

To assess the influence of using a constant static pressure at the nozzle exit plane, a separate simulation was run for the NPR = 1.88 case whose mesh within the nozzle was identical to the contraction-only simulations but also included a region in which the jet plume could develop thus allowing the domain exit boundary layer to be moved away from the nozzle exit plane (as described above). This would allow non-uniform nozzle exit static pressure profiles. The influence of the exit plane location on the development of the nozzle boundary layer could then be assessed for this NPR by comparison of the two simulations. For the simulation including a portion of the jet plume fixed static pressure boundary conditions were applied to the domain exit, far-field and free stream inlet boundaries. A further advantage of this simulations was that it allowed CFD predictions to be made for exactly the same location where measurements were taken, i.e., a small axial distance (\( \approx 0.05\text{mm} \)) downstream of the nozzle exit just within the jet plume itself; it will be seen below that this turned out to be very useful.
5.4 Predictions of Contracting Nozzle Boundary Layer Development

The following section presents results of simulations of boundary layer development through a contracting nozzle. The results for NPR = 1.88 are presented first, since an additional simulation of the nozzle and jet plume was available for this condition and is used to investigate the influence of the domain exit boundary location on the flow field before assessing the influence of the acceleration on boundary layer development. These results are then followed by the results for NPR = 1.50 and NPR = 2.40.

Figure 5.8 shows the variation of the boundary layer kinematic shape factor, kinematic skin friction coefficient, kinematic momentum thickness Reynolds number and the acceleration parameter with axial distance through the nozzle for both simulations (the domain exit located at nozzle exit or downstream to include the jet plume). The acceleration parameter used corresponds to the compressible form suggested by Nash-Webber and Oates [87] which uses fluid properties at the edge of the boundary layer and is defined in Equation 1.22. As jet plume convention dictates that axial distances are positive moving downstream, away from the nozzle exit, the nozzle exit plane is located at \( x = 0 \) and the inlet to the solution domain is at \( x \approx -0.1m \) or \( x/D \approx -1.65 \). The contraction begins at \(-0.04m \) (\( x/D = -0.66 \)).

Initially focus is placed on the simulation without the presence of a jet plume. The variation of \( H_{12i} \) shows that as the boundary layer enters the domain, the shape factor increases from the inlet value of 1.265 to a maximum of 1.397 at an axial location of \(-0.04m \), which corresponds to the beginning of the contraction. This increase in shape factor is accompanied by a large increase in the momentum thickness Reynolds number to a peak of 33184, and a decrease in the skin friction coefficient to a minimum of \( 1.445 \times 10^{-3} \). The variation of \( K_i \), which is defined using the wall static pressure gradient, shows a negative value in this region which corresponds to a mild deceleration of the flow. Inspection of the static pressure contours shown in Figure 5.9 shows a region of increasing static pressure in the vicinity of the wall at entry to the contraction. This region of higher static pressure results

<table>
<thead>
<tr>
<th>NPR 1.50</th>
<th>NPR 1.88</th>
<th>NPR 2.40</th>
</tr>
</thead>
<tbody>
<tr>
<td>LS model</td>
<td>Expt</td>
<td>LS model</td>
</tr>
<tr>
<td>( \theta_i (\times 10^{-3} m) )</td>
<td>1.39</td>
<td>1.46</td>
</tr>
<tr>
<td>( H_{12i} )</td>
<td>1.268</td>
<td>1.331</td>
</tr>
<tr>
<td>( Re_{oi} )</td>
<td>18046</td>
<td>18050</td>
</tr>
<tr>
<td>( \sigma_{f} (\times 10^{-3}) )</td>
<td>2.40</td>
<td>2.36</td>
</tr>
</tbody>
</table>

Table 5.1: Comparison of Measured and Predicted Inlet Boundary Layer Parameters

5.4 Predictions of Contracting Nozzle Boundary Layer Development

The following section presents results of simulations of boundary layer development through a contracting nozzle. The results for NPR = 1.88 are presented first, since an additional simulation of the nozzle and jet plume was available for this condition and is used to investigate the influence of the domain exit boundary location on the flow field before assessing the influence of the acceleration on boundary layer development. These results are then followed by the results for NPR = 1.50 and NPR = 2.40.

Figure 5.8 shows the variation of the boundary layer kinematic shape factor, kinematic skin friction coefficient, kinematic momentum thickness Reynolds number and the acceleration parameter with axial distance through the nozzle for both simulations (the domain exit located at nozzle exit or downstream to include the jet plume). The acceleration parameter used corresponds to the compressible form suggested by Nash-Webber and Oates [87] which uses fluid properties at the edge of the boundary layer and is defined in Equation 1.22. As jet plume convention dictates that axial distances are positive moving downstream, away from the nozzle exit, the nozzle exit plane is located at \( x = 0 \) and the inlet to the solution domain is at \( x \approx -0.1m \) or \( x/D \approx -1.65 \). The contraction begins at \(-0.04m \) (\( x/D = -0.66 \)).

Initially focus is placed on the simulation without the presence of a jet plume. The variation of \( H_{12i} \) shows that as the boundary layer enters the domain, the shape factor increases from the inlet value of 1.265 to a maximum of 1.397 at an axial location of \(-0.04m \), which corresponds to the beginning of the contraction. This increase in shape factor is accompanied by a large increase in the momentum thickness Reynolds number to a peak of 33184, and a decrease in the skin friction coefficient to a minimum of \( 1.445 \times 10^{-3} \). The variation of \( K_i \), which is defined using the wall static pressure gradient, shows a negative value in this region which corresponds to a mild deceleration of the flow. Inspection of the static pressure contours shown in Figure 5.9 shows a region of increasing static pressure in the vicinity of the wall at entry to the contraction. This region of higher static pressure results
5.4 Predictions of Contracting Nozzle Boundary Layer Development

from the requirement for the flow to start changing direction from parallel to the nozzle centreline to parallel to the nozzle wall and the associated streamline curvature requires a positive radial pressure gradient. The resultant mild deceleration of the boundary layer has the opposite effect to a favourable static pressure gradient and results in a flattening of the mean velocity profile which increases the shape factor and decreases the local skin friction coefficient [29]. The deceleration is also accompanied by a rapid thickening of the boundary layer and increased displacement ($\delta_i$) and momentum ($\theta_i$) thicknesses which is the cause of the large increase in the momentum thickness Reynolds number.

Moving further downstream, the acceleration parameter becomes positive as the boundary layer is accelerated through the convergent section. Initially, the acceleration results in the shape factor decreasing with $H_{12i}$ reaching a minimum value of 1.116 at $x = -0.012m$. Beyond $x = -0.004m$ it begins to rise very rapidly reaching a value of 1.556 at the nozzle exit. Throughout the acceleration, the skin friction coefficient has also been increasing, gradually rising in a linear manner from $2.548 \times 10^{-3}$ at the beginning of the acceleration to $4.022 \times 10^{-3}$ at $x = -0.004m$. In a similar fashion to the shape factor, downstream of this location $C_{fi}$ increases rapidly to a maximum of $6.131 \times 10^{-3}$ at the nozzle exit.

Throughout the acceleration $Re_{oi}$ has steadily dropped from its maximum value of 33184 to 365 at the nozzle exit, a decrease of two orders of magnitude, similar to the change observed in the experiments and demonstrating the domination of the thinning of the boundary layer over the increasing velocity at the edge of the boundary layer (which has doubled from 147$m/s$ at inlet to 283$m/s$ at exit). Turning attention to the variation of the acceleration parameter, $K$, (Figure 5.8(d)) it can be seen that the acceleration of the boundary layer remains fairly constant from the start of the contraction to $0.01m$ upstream of the nozzle exit with a magnitude of $K \approx 1 \times 10^{-6}$. Beyond this region the boundary layer is exposed to a very large acceleration reaching a maximum value of $10.4 \times 10^{-6}$ at the exit plane. Recalling the literature review in Chapter 1, the boundary layer is expected to begin the process of relaminarisation if the acceleration is of sufficient magnitude. Based on the analysis of Launder [75] and the findings of Back et al.[92][93] the critical value of the acceleration parameter corresponds to:

$$K \left( \frac{C_{fi}}{2} \right)^{-\frac{1}{2}} \approx 0.04$$  (5.2)

Applying this relationship to the current case implies a critical value of $K$ (based on the skin friction coefficient at inlet to the solution domain) of $1.663 \times 10^{-6}$. Based on this the boundary layer could be expected to begin to relaminarise at $x \approx -0.003m$. It is also worth noting that the relationship also implies that the critical value will increase as $Re_{oi}$ decreases through the nozzle. The distance between where the critical value is exceeded and the nozzle
exit equates to an axial distance of around twice the momentum thickness of the boundary layer at nozzle inlet. Although there is a sudden rise in the shape factor toward a laminar value, the skin friction continues to rise all the way to nozzle exit. As demonstrated in the studies of Warnack [74][89], the increase in the shape factor that follows the minimum does not guarantee the beginning of relaminarisation. An increase in the shape factor occurred for both laminarescent (still turbulent but demonstrating some laminar-like behaviour) and fully relaminarised boundary layers. In the fully relaminarised case, the increase in the shape factor was accompanied by a decrease in the local skin friction coefficient. In the study of Patel and Head [81] the increase in $H_{12}$ was seen to precede the departure of the mean velocity profile from the law of the wall which occurs during the reversion to a laminar-like state. Applying this to the current results suggests that, despite the very large favourable pressure gradient at the nozzle exit, the final stage of relaminarisation (reversion to a laminar boundary layer) has not started.

Also included in Figure 5.8 is the axial variation of the boundary layer parameters for the simulation that included the jet plume, shown as open symbols. The results show that the difference between the two different simulations is very small, with the results from the nozzle-only and nozzle-plus-plume simulations being very close. As the boundary layer becomes thinner any observed differences become less significant to the point that the results for $H_{12}$, $Re_0$, and $c_f$, become indistinguishable. Contours of static pressure for the two cases are presented in Figures 5.9 and 5.10 and axial velocity contours are presented in 5.11 and 5.12 for nozzle only and nozzle plus plume simulations respectively. The contours are markedly different only at the nozzle exit and near the centreline, therefore $M = 1$ at the nozzle lip for both cases. The effect of imposing a fixed exit static pressure is clear, leading to lower exit static pressures and contours which are normal to the centreline at the nozzle exit, compared to highly curved contours and higher than ambient static pressures when the plume is included, which are a result of the presence of a vena contracta. Comparison of profiles of mean velocity, shear stress and turbulent kinetic energy at nozzle exit (Figures 5.13- 5.15) show that, despite variations in the flow field between the two solution domains in the vicinity of the nozzle exit, the development of the boundary layer is not significantly affected (particularly the mean velocity profile), with the greatest variation occurring close to the centreline, which is beyond the region of interest. This comes as little surprise given that by the time the boundary layer has reached the exit, its thickness has reduced from $20\text{mm}$ to $\approx 0.3\text{mm}$ which means that the boundary layer is located well away from the region of the flow that is effected by the domain exit type and location. Since the experimental data demonstrated similar nozzle exit boundary layer thicknesses and kinematic momentum thickness Reynolds numbers across all of the NPRs (including supercritical NPRs) investigated, it was decided to proceed with the nozzle-only solution.
domain for all further simulations.

Attention is now turned to the axial variation of acceleration and boundary layer parameters for the \(NPR = 1.50\) case (Figure 5.16) and the underexpanded \(NPR = 2.40\) case (Figure 5.17). First impressions are that both cases follow the same trends as seen for the \(NPR = 1.88\) case, described above. This is certainly expected for the \(NPR = 2.40\) case since, beyond choking, apart from the mass flow rate (and density) variations caused by the increased static pressure levels in the nozzle, the flow field should be essentially identical to the close-to-choking case. On lowering NPR to 1.50 no substantial differences are observed, other than changes to absolute values of the maximum and minimum values of \(H_{12i}\), \(Re_{\theta i}\) and \(C_{f i}\).

The variation of \(Re_{\theta i}\) at nozzle exit is rather small with values of \(Re_{\theta i}\) of 467, 365 and 433 for \(NPR = 1.50, 1.88\) and 2.40. Noticeable is the lower value of \(Re_{\theta i}\) in the \(NPR = 1.88\), compared to the other cases, which might suggest that the influence of the acceleration in this case is greatest. To analyse the variation in values of \(Re_{\theta i}\) with NPR, the decrease can be best expressed as a ratio of the peak value of \(Re_{\theta i}\) (at inlet to the contraction) and the minimum (at nozzle exit). Recasting the results in this manner reveals that the influence of the acceleration increases with NPR with reductions in \(Re_{\theta i}\) of 55, 91 and 117 times for \(NPR = 1.50, 1.88\) and 2.40. Using the acceleration parameter as a gauge of the severity of the acceleration it can be seen that as the NPR is increased, the magnitude in the near constant region between \(-0.03m < x \leq -0.02m\) decreases (from \(1.06 \times 10^{-6}\) to \(0.69 \times 10^{-6}\) between \(NPR = 1.50\) and 2.40) but the severity of the acceleration near to the nozzle exit increases (with maximum values of 4.87, 10.4 and 11.5 \(\times 10^{-6}\) for \(NPR = 1.50, 1.88\) and 2.40). Despite the large increase in the magnitude of the acceleration parameter to values well above published critical values, the decrease in the local skin friction coefficient, which is a characteristic of the final stage of relaminarisation, is still not observed. This is believed to result from the short length of the nozzle over which the critical value is exceeded being insufficient for complete relaminarisation to occur.

To provide a better insight into the behaviour of the boundary layer as it undergoes acceleration through the nozzle, attention is turned to the development of wall-normal mean velocity and turbulence statistics profiles with axial distance. The same order of presentation of the results used as above is adopted, starting with the \(NPR = 1.88\) case, followed by the subsonic \(NPR = 1.50\) and underexpanded \(NPR = 2.40\) cases. A semi-logarithmic plot of mean velocity (n.b. velocity component parallel to the nozzle wall and not the nozzle centreline) is shown in Figure 5.18. The plots show that drastic changes to the shape of the mean velocity profile take place as the boundary layer accelerates. As the flow enters the nozzle the profiles show a developed, equilibrium behaviour (profiles 1 and 2),
5.4 Predictions of Contracting Nozzle Boundary Layer Development

following the standard logarithmic law of the wall line. A wake region is also seen. As the flow moves toward the contraction the logarithmic region of the profile can be seen to overshoot the standard log-law line with a steeper gradient, which is characteristic of a boundary layer in an adverse pressure gradient (profile 3). When the boundary layer starts its acceleration the overshoot disappears, returning briefly to following the standard log-law line once more before undershooting it at $x = -0.030m$ (profile 4). A similar undershoot of the log-law line was observed by Fernholz and Warnack [91] for their laminar cases 1 and 3. Although no complete explanation of this behaviour was offered, it was noted that the undershoot occurred at the same $Re_\theta$ and $H_{12}$, but differing values of the acceleration parameter. In the current investigation, $Re_\theta$ at this location is much higher than achieved in [91] ($Re_\theta \approx 30000$, compared to 1600). In the current investigation the static pressure gradient is not uniform across the boundary layer in the corner between the parallel and convergent sections. Figure 5.19 shows the variation of the axial static pressure gradient for the range $-0.06m \leq x \leq -0.005m$. The profiles show that at $x = -0.030m$ the pressure gradient is nearly constant across the whole of the boundary layer, but just upstream at $x = -0.040m$ the outer edge of the boundary layer has begun to accelerate whereas the near-wall region $\frac{y}{\delta} < 0.20$ is still decelerating. This will clearly cause a non-equilibrium shape to appear in the velocity profile. The results downstream of $x = -0.030m$ also show constant static pressure gradients across the whole of the boundary layer, which allows comparison with other published studies of relaminarising boundary layers where this was also observed. The exit profile, which has been omitted from Figure 5.19 to allow a suitable scale to be used, showed slightly greater acceleration in the near-wall region ($K = -2.62 \times 10^6$) compared to the outer region ($K = -2.50 \times 10^6$). In the corner the acceleration of the outer edge of the boundary layer will start earlier compared to the region close to the wall which has only just left the region of the adverse static pressure gradient observed in the wall region at inlet to the contraction. Therefore the observed undershoot of the profiles near the corner can be attributed to the non-uniform favourable pressure gradient and hence acceleration of the boundary layer.

Between $x = -0.03m$ and $x = -0.005m$ the velocity profiles (profiles 5 - 7) can be seen to overshoot the log-law. This is a characteristic shown by zero pressure gradient boundary layers in the last stages of natural transition from laminar to turbulent which have not reached equilibrium [67]. A similar overshoot was seen in the low Reynolds number, fully developed turbulent pipe flow predictions used for validation of the Launder-Sharma model in Section 3.8. The extent of the logarithmic region is much reduced compared to the inlet profile and is related to the Reynolds number [67]. Close to the nozzle exit it is argued that no logarithmic region is present. It is worth noting that the non-dimensional velocities are seen to reduce beyond $y^+ \geq 10^4$. This shows that the acceleration of the region of the flow
5.4 Predictions of Contracting Nozzle Boundary Layer Development

The final curve (profile 8) corresponds to the nozzle exit boundary layer. At this location the profile lies entirely beneath the log-law line and corresponds to the region of very rapid flow acceleration at the nozzle exit (see Figure 5.8(d)). Close inspection of the profile does not reveal any clear logarithmic region but also does not demonstrate a laminar profile in the outer region of the boundary layer and, therefore, the boundary layer is still turbulent, although highly non-equilibrium. It is worth noting that the Reynolds number of the boundary layer at this location \(Re_{ol} = 365\) is approaching the limit at which a turbulent boundary layer can exist. The review of incompressible boundary layer data by Fernholz and Finley [67] identified logarithmic regions at Reynolds numbers as low as 350. Similar behaviour of the mean velocity profiles was exhibited in the results of the relaminarisation study of Patel and Head [81] where the logarithmic region had all but disappeared and the whole profile lay below the standard log-law line. This occurred just before the overshoot of the log-law line and the decrease in the local skin friction began, both of which are important characteristics of the final stages of relaminarisation, and are not observed in the present data.

The development of the wall normal profiles of Reynolds shear stress \(\overline{\rho u'v'}\) (made non-dimensional with respect to \(\overline{\rho u'^2}\)) with axial distance is shown in the semi-logarithmic plot in Figure 5.20. Note that, unlike the mean velocity profiles above, the Reynolds shear stress has not been resolved normal to the wall and is still in Cartesian form (orthogonal to the nozzle centreline). Although absolute values of the Reynolds stresses in the near-wall region are expected to rise in the initial stages of the acceleration [78][84][91] as the flow adjusts to the increase in the wall shear stress, the use of inner-wall scaling helps to identify the degree to which the internal turbulent shear stress lags behind the wall shear stress. Recalling the results of the validation of the Launder-Sharma model for relaminarising boundary layers in Section 3.8, care is needed in interpretation of predicted Reynolds stress behaviour as the model was shown to over-predict the suppression of the shear stress. The model did, however, capture the correct trends, thus allowing the present predictions to be used to assess the effects of the acceleration and to compare the different NPRs simulated.

The profiles for NPR = 1.88 (Figure 5.20) show that at the first location \((x = -0.06m)\), the boundary layer demonstrates behaviour similar to a zero pressure gradient boundary layer, with a peak value of \(\overline{\rho u'v'} \approx 0.90\) occurring at the edge of the buffer region (at \(y^+ \approx 10^2\)) and then decreasing to zero at the outer edge (\(y^+ \approx 10^4\)). As the boundary layer encounters the adverse pressure gradient, there is a large increase to a value of 2.05, a result of the reduced wall shear stress and hence the value of \(\overline{u'v'}\) which is used to non-dimensionalise the results. The profile also demonstrates a shift in the location of the peak value away from
the wall. Beyond the deceleration (profile 4), the profile in the inner region quickly returns to the shape seen prior to the adverse pressure gradient but a second peak still remains toward the outer edge, showing the slower recovery of the outer region of the boundary layer. Of prime interest in the current investigation is the behaviour of the Reynolds shear stress as the boundary layer is accelerated. Beyond \( x = -0.040 \) (profiles 5 - 8) there is a rapid decrease in the shear stress; initially the greatest decrease occurs in the near-wall region, but for \(-0.02m < x < -0.005m\) (prior to the very rapid acceleration at the nozzle exit) the profiles adopt a saddle-like appearance, with a peak of 0.5 times that seen in a zero pressure gradient boundary layer at \( y^+ \approx 20 \), which decreases slowly with downstream distance and an outer peak which decreases more rapidly. The greatest changes are seen at the nozzle exit (profiles 7 and 8) where peak values have decreased to 0.146.

Figure 5.21 presents a similar illustration of the turbulent kinetic energy development (non-dimensionalised with respect to \( \mu u_i^2 \)) again shown in semi-logarithmic form. The results follow a similar trend to the Reynolds shear stress described above. Starting with the profile at \(-0.06m\), a region where the boundary layer is shown to possess ZPG boundary layer characteristics, the profile has a peak value of 3.15 at \( y^+ \approx 20 \). The peak rapidly increases in the region of the adverse pressure gradient, for the same reason as explained above. As the boundary layer accelerates, the near-wall peak decreases with axial distance but remains at the same non-dimensional distance from the wall. The greatest reduction in the non-dimensional turbulence level is seen in the outer region of the boundary layer where values decreased to less than a third of the inlet values. There are again large changes in the profile in the region of very severe acceleration in the last few \( mm \) of the nozzle, where the near-wall peak reduces to a value of 0.85 and the outer peak to 0.42.

The influence of NPR on the development of the nozzle boundary layer is now investigated. As the development of the boundary layer up to \( x \approx -0.01m \) was remarkably similar for all NPRs only three locations are considered for each NPR, namely \(-0.01m, -0.005m\) and nozzle exit. Figure 5.22 compares mean velocity profiles in semi-logarithmic form. The data show little difference between NPRs at \(-0.01m\) and \(-0.005m\) for all NPRs. At nozzle exit, however, variation is observed; profiles show increased undershoot of the standard logarithmic law of the wall as NPR increases. This is consistent with the increase of values of K (above published critical values) close to the nozzle exit with increasing NPR, as described above. The greatest change occurs between \( NPR = 1.50 \) and 1.88. Little difference is seen between near-choked and supercritical NPRs, which is expected, since once the nozzle has choked the internal flow field is essentially invariant. Figure 5.23 compares the variation of shear stress profiles with NPR. Like the mean velocity profiles, little variation occurs until the last \( 5mm \) where peak values rapidly decrease. Most noticeable is the close agreement between exit profiles for \( NPR = 1.88 \) and \( NPR = 2.40 \) which are both approximately half
5.4 Predictions of Contracting Nozzle Boundary Layer Development

that of the \( NPR = 1.50 \) case (\( \approx 0.15U_g^2 \) compared to \( 0.30U_g^2 \)). The trends seen in the shear stress profiles are mirrored in the turbulent kinetic energy profiles in Figure 5.24, close agreement between \( NPR = 1.88 \) and \( NPR = 2.40 \) cases, and greater reduction in peak values at the nozzle exit compared to \( NPR = 1.50 \).

Before comparing experimental and numerical results it is prudent to investigate to what extent performing boundary layer measurements just outside of the nozzle influences the measured profile and derived boundary layer parameters such as kinematic momentum thickness and shape factor. Although no internal nozzle exit measurements were conducted for reasons outlined in Chapter 2, the numerical simulation of the nozzle with initial jet plume region can provide valuable insight into the choice of measurement location and the applicability of the work of Morris and Foss [104] to the current investigation.

Figures 5.25 and 5.26 show comparisons of a nozzle exit boundary layer velocity profile extracted from the nozzle-plus-plume simulation at the nozzle exit and the same axial location downstream of the nozzle lip as the experimental measurement traverse plane. Figure 5.25 shows the same two profiles non-dimensionalised with respect to the kinematic displacement thickness and edge velocity. The profile taken from downstream of the nozzle lip (denoted External) exhibits a flatter, more laminar-like appearance compared to the profile within the nozzle at the exit (denoted Internal) with velocity increasing more gradually and linearly with distance up to \( y/\delta_i \approx 2 \) and overshooting the internal profile beyond \( y/\delta_i \approx 2 \). The difference in the near wall region is further highlighted by the Clauser plot in Figure 5.26. Also included in the figure is the non-dimensional distance corresponding to \( 2\theta_i \) (\( \theta_i \) taken at the nozzle exit plane) which was the lower limit where Morris and Foss [104] claimed a velocity profile measured just downstream of the nozzle lip would be identical to a profile at the nozzle lip itself. The region \( 0 < y < 2\theta_i \) has been influenced by the wall boundary layer to free shear layer change. The profiles show good agreement between internal and external profiles for distances greater than \( 2\theta_i \), in agreement with Morris and Foss [104] but show a significant overshoot by the nozzle exit profile in the region \( y/\delta_i < 2 \). As it is the region close to the wall that provides the greatest contribution to the integral parameters \( \delta_i^* \) and \( \theta_i \), changes in the shape of the velocity profile in the region \( y < 2\delta_i \) will make a significant difference to global boundary layer parameters derived from the internal and external profiles, as shown in Table 5.2. The results show that the External profile yields lower \( \theta_i \) and \( Re\theta_i \) and a higher, more laminar \( H_{12i} \) as illustrated by Figure 5.25.

It is immediately apparent from the above results that, as a result of differing axial locations, direct comparison of predicted and measured nozzle exit boundary layers could only be made for the \( NPR = 1.88 \) case, since the nozzle plus plume simulation allows a 'like with like' comparison and proper assessment of the turbulence model. Since time did not
5.4 Predictions of Contracting Nozzle Boundary Layer Development

<table>
<thead>
<tr>
<th></th>
<th>External</th>
<th>Internal</th>
</tr>
</thead>
<tbody>
<tr>
<td>$Re_{\theta_i}$</td>
<td>640</td>
<td>685</td>
</tr>
<tr>
<td>$\theta_i (\pm 10^{-5} \text{m})$</td>
<td>2.629</td>
<td>2.969</td>
</tr>
<tr>
<td>$H_{12i}$</td>
<td>1.918</td>
<td>1.555</td>
</tr>
</tbody>
</table>

Table 5.2: Comparison of Boundary Layer Parameters from Internal and External Measurement Planes

permit nozzle plus plume simulations for the other NPRs, in order to compare results in the absence of plume simulations for all NPRs investigated numerically a method of correction was required to allow back to back computational-experimental comparison. A correction factor was derived from the non-dimensional profiles shown in Figure 5.26 for $NPR = 1.88$. The correction factor was the ratio at each value of $y$ in the free-shear-affected region of the velocity at the external location to the velocity predicted at the nozzle exit plane itself. It was assumed as a first approximation that this correction factor would be the same for other NPRs. The corrected profiles for all NPRs were then used to derive global boundary layer parameters to compare with the experimental results.

Figures 5.27, 5.28 and 5.29 compare internal, external (corrected for $NPR = 1.50$ and $2.40$ and directly predicted for $NPR = 1.88$) and experimental profiles for all NPRs. The $2\theta_i$ limit is also included (based on internal CFD results). The profiles show all internal profiles overshoot the experimental data for $y/\theta_i < 2$ as expected based on the comparison of predicted internal and external profiles in Figure 5.25. The profiles agree closely with the experimental data for $y/\theta_i > 2$ as suggested by Morris and Foss [104] and the corrected profiles agree quite well for $y/\theta_i < 2$, which shows that the turbulence model is capable of predicting highly accelerated nozzle boundary layer velocity profiles. The $NPR = 2.40$ profile shows a less rapid flattening of the profile towards the outer edge of the boundary layer. This variation is not unexpected due the presence of the Prandtl-Meyer expansion fan starting at the nozzle lip.

Table 5.3 compares experimental and corrected CFD global boundary layer parameters. Although $Re_{\theta_i}$ is underpredicted for $NPR = 1.50$, generally good agreement between numerical simulation and experiment is observed particularly for $NPR = 1.88$ and $NPR = 2.40$ with $H_{12i}$ approaching the more laminar values seen experimentally.

Figures 5.30 and 5.31 show comparison between measured and predicted nozzle exit turbulent kinetic energy and shear stress for all three NPRs. Experimental values of $k$ were defined as:

$$k = \frac{1}{2} (u'^2 + 2v'^2)$$

(5.3)
5.4 Predictions of Contracting Nozzle Boundary Layer Development

<table>
<thead>
<tr>
<th></th>
<th>NPR 1.50</th>
<th></th>
<th>NPR 1.88</th>
<th></th>
<th>NPR 2.40</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>LS model</td>
<td>Expt</td>
<td>LS model</td>
<td>Expt</td>
<td>LS model</td>
<td>Expt</td>
</tr>
<tr>
<td>$\theta_i (\pm 10^{-5} m)$</td>
<td>2.607</td>
<td>2.500</td>
<td>1.488</td>
<td>1.578</td>
<td>1.379</td>
<td>1.670</td>
</tr>
<tr>
<td>$H_{12i}$</td>
<td>1.89</td>
<td>2.08</td>
<td>1.91</td>
<td>2.06</td>
<td>1.85</td>
<td>2.20</td>
</tr>
<tr>
<td>$Re_{\theta i}$</td>
<td>329</td>
<td>434</td>
<td>433</td>
<td>415</td>
<td>484</td>
<td>469</td>
</tr>
</tbody>
</table>

Table 5.3: Comparison of Measured and Predicted Nozzle Exit Boundary Layer Parameters for The Contracting Nozzle

As only two velocity components could be measured ($U$ and $V$) it was assumed that $w'^2 = v'^2$. The data show the predictions fail to capture correctly the near-wall behaviour of $k$ or the shear stress for all NPRs; most noticeable is the absence of a large near-wall peak. Neither result is unexpected as the predictions of the relaminarising boundary layer in Chapter 3 showed the turbulence model failing to predict correctly the near-wall peak in $k$ upstream of the acceleration (Figure 3.7) and the overprediction of the suppression of the Reynolds shear stress once the boundary layer was accelerated (Figure 3.13). Numerical studies of relaminarisation in the current literature have not presented results of turbulence prediction, which would allow a more thorough comparison.

The results from the three test cases have confirmed the experimental observations by clearly showing that the nozzle exit boundary layers are highly influenced by the favourable static pressure gradients that exist within the nozzle and details of the development of the mean flow within the nozzle have been examined using the Launder-Sharma low Reynolds number $k - \epsilon$ turbulence model. The influence of the large favourable pressure gradient is particularly characterised by the significant reductions in $Re_{\theta i}$ (two orders of magnitude), increased skin friction, and non-equilibrium, transitional-like behaviour of the profiles of non-dimensional mean velocity (overshoot of the standard log-law line, reduction of the extent of the logarithmic region and the disappearance of the wake region) and large reductions in the non-dimensional Reynolds shear stress and turbulent kinetic energy. The profiles and boundary layer parameters have shown that for the range of NPRs investigated that the changes in boundary layer behaviour that occur up to the last 5mm of the flow appear to be identical, with the exception of the $Re_{\theta i}$ value at the inlet, which is of course a function of the flow conditions within the delivery duct. The only significant variations with NPR occur in the last 5mm of the nozzle where the negative axial pressure gradient and, hence, the magnitude of the acceleration increase very rapidly, with the acceleration increasing with NPR. In this region published critical values of the acceleration parameter are greatly exceeded. It is clear that the boundary layer begins to revert to a laminar-like state, but it appears that the distance over which the high levels of the acceleration parameter are sustained are too short for complete reversion to occur. In the relaminarising cases of Warnack and
5.5 Predictions of Nozzles with Parallel Extension

Fernholz [74], the acceleration is maintained at levels of the acceleration parameter which will involve relaminarisation for distances of $240\theta_s$ and $170\theta_s$ for cases 2 and 4, respectively (where $s$ denotes the start of the acceleration); in the compressible nozzle studies of Back et al. [92] the boundary layer was exposed to a sustained acceleration for $70\theta$ (note Back et al. used the compressible definitions of boundary layer integral parameters). In the study of relaminarising compressible nozzle boundary layers by Nash-Webber and Oates [87] the acceleration was maintained for up to 16 inches ($\approx 420mm$), which corresponds to approximately $100\theta_s$. In the current investigation, the distance over which the high values of the acceleration parameter are achieved are approximately $3 - 4\theta_i$ which is insufficient for the relaminarisation process to proceed very far. It is, therefore, argued that in the current nozzles the boundary layers remain laminarescent (demonstrating some laminar characteristics, but still turbulent) at the exit.

5.5 Prediction of Boundary Layer Development in Nozzles with Parallel Wall Extension

In the previous chapter, it was shown experimentally that the addition of a parallel wall nozzle extension piece to the end of the contraction-only nozzles resulted in large changes in the shape and thickness of the nozzle exit boundary layer. Most noticeable was the order of magnitude increase in the kinematic momentum thickness Reynolds number. The changes in the nozzle exit conditions, both in terms of the boundary layer and the vena contracta were also shown to influence the development of the jet plume shear layer and the inviscid core shock structures when operating at supercritical NPRs. The following section uses CFD predictions to investigate the mechanisms responsible for these observed changes using the LU60(P) nozzle operating at the same NPRs as used in the previous section. To allow easier comparison with the contraction-only nozzle results, the same co-ordinate origin is used, with the end of the contraction corresponding to an axial location of $x = 0m$ and, therefore, the nozzle exit plane corresponds to $x = +0.0318m$ for the nozzle with extension piece. The results are presented in order of increasing NPR in terms of the development of the boundary layer parameters $H_{12i}$ and $Re_{\theta_i}$ and the development of the profiles of mean velocity and turbulence statistics.

Figures 5.32 to 5.34 show the development of $H_{12i}$ and $Re_{\theta_i}$ with axial distance for $NPR = 1.50$, $NPR = 1.88$ and $NPR = 2.40$, respectively. Focusing on the $NPR = 1.50$ case, the results show the recovery of the turbulent boundary layer towards an equilibrium zero pressure gradient state after acceleration through the contraction. $H_{12i}$ shows a large decrease as the boundary layer moves into the parallel wall section and the shape of the
5.5 Predictions of Nozzles with Parallel Extension

boundary layer gradually changes from a transitional shape back to a turbulent one. Beyond \( x = 0.02m \), \( H_{12i} \) reaches a constant value of 1.18 which is significantly lower than that of the equilibrium boundary layer at the nozzle inlet (1.268). After entry to the parallel section there is a gradual rise in \( \Re_{\theta} \), increasing from the minimum value of 467 at the end of the contraction to 6885 at the end of the nozzle. This increase in the Reynolds number can be seen to occur in two stages, a more rapid rise corresponding to the region where the shape factor is dropping (\( 0m \leq x \leq 0.02m \)) followed by a more gradual rise through the remainder of the nozzle. Similar changes to the boundary layer parameters also occur in the \( \text{NPR} = 1.88 \) and \( \text{NPR} = 2.40 \) cases (Figures 5.33 and 5.34), but the changes are more sudden. For the \( \text{NPR} = 1.88 \) case \( H_{12i} \) continues to rise beyond the end of the contraction region, reaching a value of 1.8 at \( x = 0.005m \), but by \( x = 0.010m \) it has dropped to the turbulent value of 1.17, a much more rapid change than observed in the \( \text{NPR} = 1.50 \) case. The Reynolds number also sees a similar rapid increase, from the minimum value at the exit of 365 to 6700 by \( x = 0.015 \), gradually increasing to 6923 at the exit. At the underexpanded \( \text{NPR} \) of 2.40, similar behaviour to the near-choked \( \text{NPR} = 1.88 \) case is seen, with the shape factor continuing to grow beyond the end of the contraction, followed by a very rapid decrease in \( H_{12i} \) and increase in \( \Re_{\theta} \) to 8810 at the nozzle exit.

Turning attention to the development of the predicted mean velocity profiles, the semi-logarithmic plot in Figure 5.35 shows the mean velocity profiles, scaled in inner wall units. The results show a very rapid recovery of the near-wall velocity distribution, compared to the profile at the exit of the contraction region. At the location of the first profile, \( x = 0.005m \) (only 15% of the parallel section length), a clear logarithmic region has already reappeared although the profile departs from the \( u^+ = y^+ \) line prematurely at \( y^+ \approx 8 \) and undershoots the standard log line. By \( x = 0.010m \), the extent of the logarithmic region has extended considerably and now lies on the standard log-law line. Subsequent profiles collapse onto each other and show a slight overshoot of the log-law line in a similar fashion to the results of the fully developed, low Reynolds number pipe flow used to validate the model in Section 3.8.

To assess the recovery of the outer region of the boundary layer, the mean velocity profiles are presented in outer-wall scaling (Section 1.5.2) in Figure 5.36. Also plotted is the universal logarithmic velocity distribution for compressible boundary layers as given by Fernholz and Finley [67]. The profiles show large variations at the first two locations which correspond to where rapid changes in the shape factor and Reynolds number are occurring, followed by a collapse of the profiles for all subsequent locations. At all of the axial locations investigated the profiles undershoot the logarithmic line for a zero pressure gradient boundary layer substantially, as seen also by Fernholz and Warnack [91], suggesting that the length of the parallel section is long enough to recover a thick, turbulent boundary layer with near-wall
equilibrium but is of insufficient length for the outer region to recover fully to an equilibrium condition.

Profiles of the non-dimensional Reynolds shear stress for $NPR = 1.50$ (Figure 5.37) also show a rapid recovery in the near-wall region. The profiles at $x = 0.005m$ and $x = 0.010m$ show peak values of 1.5 and 1.13, respectively, which is indicative of the low values of wall shear stress (as seen in the previous section at inlet to the contraction) and will be discussed in more detail below. Beyond $x = 0.010m$ a peak value of 0.9 is maintained, which is similar to the value quoted in the review of experimental data of Fernholz and Finley [67] which showed a plateau in the data corresponding to a non-dimensional value between 0.92 and 0.95. The results of Figure 5.37 also show a second plateau at the outer edge of the boundary layer ($y^+ > 500$) which suggests the outer part of the boundary layer has not fully recovered to a zero-pressure-gradient-like behaviour and lags the recovery of the near-wall region. The same behaviour is also observed in the Reynolds shear stress results for the high NPR cases. The study of the laminarercial boundary layers by Fernholz and Warnack [91] showed that once the acceleration was relaxed, the near-wall region quickly returned to a pre-acceleration equilibrium state. The authors also showed that the outer region was much slower to respond to the removal of the favourable pressure gradient and returned to an equilibrium state at a much slower rate than the near-wall region.

The velocity profiles for the two larger NPR cases show similar behaviour so only the higher $NPR = 2.4$ case is given in Figure 5.38, to illustrate the fact that the premature departure from the linear law and undershoot of the standard log-law line at the first location just inside the parallel extension is present again, even stronger than at $NPR = 1.5$. This is clearly a response to the pressure gradients induced at the discontinuity in slope between convergent and parallel walls as shown below.

The turbulent shear stress and outer-wall-law-scaled velocity profiles for the higher NPR cases are essentially similar to the $NPR = 1.50$ case. For completeness Figures 5.39 and 5.40 show outer-wall law scaled and shear stress predictions for the highest NPR (2.4). The shear stress behaviour is, in trend terms, identical to the $NPR = 1.50$ case, although the peak shear stress value in the early part of the parallel section is much higher (2.5 compared to 1.5) which is testimony to the strong pressure gradients induced by the corner (to be discussed below). The rapid recovery to the two-layer structure of an inner region near to equilibrium and an outer region lagging behind is again seen. The outer wall scaling profiles (Figure 5.39) show that the profile at the first location is sufficiently distorted to overshoot the log-law line, but elsewhere the behaviour is similar.

As one of the aims of this thesis is to assess the ability of the turbulence model to predict nozzle boundary layer development, comparison between measured and predicted
5.5 Predictions of Nozzles with Parallel Extension

Table 5.4: Comparison of Measured and Predicted Boundary Layer Parameters for Nozzle With Parallel Wall Extension

<table>
<thead>
<tr>
<th></th>
<th>NPR 1.50</th>
<th></th>
<th>NPR 1.88</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>LS model</td>
<td>Expt</td>
<td>LS model</td>
<td>Expt</td>
</tr>
<tr>
<td>$\theta_i (\times 10^{-4} m)$</td>
<td>3.48</td>
<td>3.62</td>
<td>2.55</td>
<td>2.39</td>
</tr>
<tr>
<td>$H_{12i}$</td>
<td>1.18</td>
<td>1.22</td>
<td>1.17</td>
<td>1.26</td>
</tr>
<tr>
<td>$Re_{\theta_i}$</td>
<td>6885</td>
<td>7494</td>
<td>6923</td>
<td>6852</td>
</tr>
</tbody>
</table>

boundary layer velocity profiles at nozzle exit is now made. Comparisons are limited to the two subsonic cases since the measured profile for the underexpanded case is highly influenced by the Prandtl-Meyer expansion fan that exists at the measurement location, as shown by Figure 4.70. Unlike the contraction-only nozzle simulations, it was not felt that correction of the predicted profiles was required in order to compare simulation with experiment.

Table 5.4 compares measured and predicted global boundary layer parameters. The values show that the model predicts a similar recovery of the momentum thickness and kinematic shape factor from the end of the contraction as observed in the experimental data. Non-dimensional mean velocity profiles are presented in Figure 5.41. In a similar manner to the comparison of the uncorrected exit profiles for the contraction only nozzle, the predicted values overshoot the experimental profile at the lower end of the scale (although the degree of overshoot is greatly reduced) but show good agreement toward the outer edge of the boundary layer where the flow has not fully recovered. Figures 5.42 and 5.43 compare non-dimensional profiles of measured and predicted turbulent kinetic energy and shear stress for $NPR = 1.50$ ($NPR = 1.88$ results were very similar). The data show that near-wall peak values of $k$ are greatly underpredicted by the Launder-Sharma turbulence model, predicted values being approximately one third of measured values. The same underprediction of near-wall peak values is also seen in the profiles of the Reynolds shear stress where predicted peak values are approximately 25% of the measured peak value.

Also of interest is the influence that the addition of the parallel extension has on the vena contracta. The experimental jet plume measurements, presented in Chapter 2, showed the disappearance of any obvious vena contracta effects after the addition of the extension pieces (e.g. the absence of any region of increasing centreline velocity beyond the exit of the nozzle). To investigate this and to provide better insight into the changes in the state and thickness of the boundary layer, Figures 5.44-5.48 present contours of the static pressure and mean axial velocity or Mach number fields for all three NPRs.

Starting with $NPR = 1.50$, the static pressure contours (Figure 5.44) show a region of localised low static pressure has formed immediately downstream of the end of the contraction
5.5 Predictions of Nozzles with Parallel Extension

region, again required to balance streamline curvature. The contours of axial velocity in Figure 5.45 and the superimposed particle traces show that in the region of adverse pressure gradient, the boundary layer has separated from the nozzle wall and a thin recirculation region has formed. The predicted length of the thin recirculation bubble starts at the end of the contraction and extends to $x = 0.00135m$ (based on wall shear stress). The location of the first profiles in the plots of mean velocity and Reynolds shear stress correspond to an axial distance of $x = 0.005m$ (i.e. well downstream of this region). The recirculation helps to explain the overshoot of the non-dimensional shear stress profiles and the undershoot of the standard log-law line at this location as the result of the boundary layer recovering from the effects of the separation and reattachment. Similar behaviour was seen downstream of boundary layer reattachment in the study of turbulent boundary layer separation by Song and Eaton [155]. The region of low static pressure results from the continued acceleration of the flow as it leaves the contraction. If the extension were not present (i.e. in the jet plume), the cross section of the flow would continue to decrease beyond the nozzle exit and ambient flow would move in to fill the void. When the solid walls of the nozzle are present, and no fluid can enter to fill the void, a low pressure region forms which then induces an adverse pressure gradient region near the wall and hence a recirculation.

The flow behaviour in this region changes as NPR is increased. At $NPR = 1.88$, the static pressure and Mach number contours in Figures 5.46 and 5.47 show that the decrease in the static pressure at the end of the contraction region is of sufficient magnitude that the local total-to-static pressure ratio exceeds the critical value of 1.893 and a supersonic region (with a peak Mach number of 1.105) is formed, which is terminated by a shock wave at $x = 0.0036m$. As shown by the particle traces, a small recirculation region is again predicted, extending from $x = 0.000104m$ to the location of the shock wave, but is now considerably thinner than observed in the $NPR = 1.50$ case. Comparing the static pressure contours of the $NPR = 1.55$ and 1.88 cases, the persistence of the vena contracta is clearly significantly reduced with little variation shown beyond $x = 0.011m$. Increasing the NPR to 2.40 shows the same formation of a region of supersonic flow (Figure 5.49). The peak Mach number has increased slightly to 1.12 but the shock occurs at the same axial location. Unlike at the lower NPRs, no recirculation region was predicted to form at $NPR = 2.40$. The disappearance of the vena contracta is even more apparent in the pressure profiles at $NPR = 2.40$, where the static pressure contours can be seen to be normal to the nozzle wall beyond the location of the shock (Figure 5.48). The effect of the parallel-wall section on the vena contracta is also highlighted in the plot of static pressure variation along the centreline in Figure 5.50. Values have here been non-dimensionalised by nozzle exit static pressure. The curves show the continued rapid decrease of the static pressure at the centreline beyond the end of the contraction as the flow continues to accelerate due to the contraction but slows down around
\[ \approx 0.007 m. \]

### 5.6 Closure

A numerical approach to predicting boundary layer development through conical nozzles with and without parallel wall extensions using the Launder-Sharma low Reynolds number \( k - \epsilon \) turbulence model has been presented in this Chapter. Solution domain inlet conditions were generated through the use of precursor calculations so that realistic nozzle inlet conditions could be used. The effect of locating the domain exit at the nozzle exit plane to avoid the necessity of computing the jet plume and associated computational overheads was addressed. Computations with and without a jet plume region revealed that, despite large variations in the flow field towards the nozzle exit and centreline, for the highly suppressed boundary layers that develop within the nozzle's contraction, the predicted development of the boundary layer is essentially unaltered by the choice of the domain exit location.

The Launder-Sharma model was then used to assess the influence of the favourable pressure gradient and NPR on boundary layer development and showed that acceleration significantly influences the boundary layer. The results showed that variations in the NPR had little effect on the development of the boundary layer in the first 90% of the nozzle's length but significantly influenced the last 10% where magnitudes of the acceleration parameter, \( K \), were seen to increase with NPR. Despite magnitudes of the acceleration exceeding published critical values, it was shown that the duration of the acceleration is insufficient for the boundary layer to undergo the process of complete relaminarisation. The addition of parallel wall extension pieces to the end of the nozzle contraction to allow a boundary layer relaxation region were also simulated to investigate the boundary layer recovery which had been observed experimentally. The extensions were shown to influence significantly the shape and thickness of the nozzle exit boundary layer and cause the formation of a region of recirculating flow downstream of the contraction and localised supersonic flow for high subsonic and underexpanded NPRs. The extensions were also shown to remove the continued acceleration of the flow beyond the end of the nozzle, the vena contracta, which had been shown in the previous chapter to influence the development of the near-field region of the jet plume and the distribution of shock cells in underexpanded jet flows. Comparison with LDA measurements of the nozzle exit boundary layer showed that the turbulence model was capable of predicting the development of the boundary layer global parameters and mean velocity profile within both types of nozzle geometry with reasonable accuracy but failed to capture the development of the turbulence statistics.
5.7 Figures
Figure 5.1: LU60 Nozzle Geometry

Figure 5.2: LU60 Nozzle With Parallel-Walled Extension Geometry
Figure 5.3: LU60 Nozzle With Jet Plume Region Geometry

Figure 5.4: LU60 Mesh
Figure 5.5: LU60 Nozzle With Parallel-Walled Extension Mesh

Figure 5.6: LU60 Nozzle With Jet Plume Mesh
Figure 5.7: Inlet Profiles for $NPR = 1.50$
Figure 5.8: Variation of Boundary Layer Parameters with Axial Location for NPR=1.88
Figure 5.9: Contours of Static Pressure for Prediction without Jet Plume (NPR=1.88)

Figure 5.10: Contours of Static Pressure for Prediction with Jet Plume (NPR=1.88)
Figure 5.11: Contours of Axial Velocity for Prediction without Jet Plume (NPR=1.88)

Figure 5.12: Contours of Axial Velocity for Prediction with Jet Plume (NPR=1.88)
Figure 5.13: Comparison of Mean Velocity Profiles

Figure 5.14: Comparison of Reynolds Shear Stress ($\overline{p u' v'}$) Profiles
Figure 5.15: Comparison of Turbulent Kinetic Energy Profiles
Figure 5.16: Variation of Boundary Layer Parameters with Axial Location for NPR=1.50

(a) Kinematic Shape Factor
(b) Local Skin Friction
(c) Kinematic Momentum Thickness Reynolds Number
(d) Acceleration Parameter
Figure 5.17: Variation of Boundary Layer Parameters with Axial Location for NPR=2.40.

(a) Kinematic Shape Factor

(b) Local Skin Friction

(c) Kinematic Momentum Thickness Reynolds Number

(d) Acceleration Parameter
Figure 5.18: Variation of Mean Velocity Profiles for NPR 1.88
Figure 5.19: Variation of Static Pressure Gradient Profiles for NPR 1.88

Figure 5.20: Variation of Reynolds Shear Stress $\bar{\rho}u'\bar{v}'$ Profiles for NPR 1.88
Figure 5.21: Variation of Turbulent Kinetic Energy Profiles for NPR=1.88
Figure 5.22: Comparison of Mean Velocity Profiles for Varying NPR
Figure 5.23: Comparison of Reynolds Shear Stress $\overline{\nu u'}$ Profiles for Varying NPR
Figure 5.24: Comparison of Turbulent Kinetic Profiles for Varying NPR
Figure 5.25: Comparison of Internal and External Predicted Velocity Profiles \((NPR = 1.88)\)

Figure 5.26: Comparison of Internal and External Predicted Velocity Profiles (Clauser Plot) \((NPR = 1.88)\)
Figure 5.27: Comparison of Experimental and Predicted Mean Velocity Profiles ($NPR = 1.50$)

Figure 5.28: Comparison of Experimental and Predicted Mean Velocity Profiles ($NPR = 1.88$)
Figure 5.29: Comparison of Experimental and Predicted Mean Velocity Profiles (NPR = 2.40)
Figure 5.30: Comparison of Measured and Predicted Nozzle Exit Turbulent Kinetic Energy Profiles
Figure 5.31: Comparison of Measured and Predicted Nozzle Exit Reynolds Shear Stress Profiles
Figure 5.32: Variation of Boundary Layer Parameters Within Parallel Nozzle Extension for NPR=1.50

Figure 5.33: Variation of Boundary Layer Parameters Within Parallel Nozzle Extension for NPR=1.88
Figure 5.34: Variation of Boundary Layer Parameters Within Parallel Nozzle Extension for NPR=2.40

Figure 5.35: Recovery of near-wall Region Mean Velocity Profile for NPR=1.50
Figure 5.36: Recovery of Outer Region of the Mean Velocity Profile for NPR=1.50

Figure 5.37: Recovery of the Reynolds Shear Stress $\bar{\rho}u'v'$ Profile for NPR=1.50
Figure 5.38: Recovery of near-wall Region Mean Velocity Profile for NPR=2.40

Figure 5.39: Recovery of Outer Region of the Mean Velocity Profile for NPR=2.40
Figure 5.40: Recovery of the Reynolds Shear Stress $\overline{u'v'}$ Profile for NPR=2.40

Figure 5.41: Comparison of Predicted and Measured Exit Boundary Layers for Nozzles with Parallel Extensions
Figure 5.42: Comparison of Predicted and Measured Turbulent Kinetic Energy Profiles ($NPR = 1.50$)

Figure 5.43: Comparison of Predicted and Measured Reynolds Shear Stress Profiles ($NPR = 1.50$)
Figure 5.44: Contours of Static Pressure in the Nozzle Parallel Extension for NPR=1.50

Figure 5.45: Contours of Axial Velocity Near the Wall in the Parallel Extension for NPR=1.50
Figure 5.46: Contours of Static Pressure in the Nozzle and Parallel Extension for NPR=1.88

Figure 5.47: Contours of Mach Number Near the Wall in the Parallel Extension for NPR=1.88
Figure 5.48: Contours of Static Pressure in the Nozzle and Parallel Extension for NPR=2.4

Figure 5.49: Contours of Mach Number in the Nozzle and Parallel Extension for NPR=2.4
Figure 5.50: Parallel Extension Centreline Static Pressure Variations
Chapter 6

Conclusion and Further Work

6.1 Summary and Conclusions

The overall aim of this thesis, as outlined in Chapter 1, was to assess the influence of nozzle scale and operating conditions on nozzle exit boundary layers and the initial development of the jet plume for high subsonic and supersonic (underexpanded) NPRs, which had been identified as an area requiring further work by the review of current literature. This was achieved through a combined experimental and computational approach, charting the evolution of the boundary layer starting from inlet to the nozzle, following its passage through the nozzle and its separation from the nozzle exit lip to form the jet shear layer.

The experimental investigation of nozzle inlet and exit boundary layers was achieved using a combination of intrusive (pneumatic probe) and non-intrusive (LDA) measurement techniques. The data revealed that a major effect of the acceleration on the boundary layer within conical nozzles was the dramatic reduction in its thickness and Reynolds number accompanied by a change in its shape towards a more laminar-like structure. The influence of the favourable pressure gradient was shown to be affected by both scale and NPR, with greater reduction in $Re_{oi}$ occurring for the larger nozzle scale (which had the shortest contraction length) and larger NPRs. Investigation of geometric influences on the nozzle exit boundary layer was achieved through the addition of parallel wall extensions to the end of the contraction-only nozzles which added a relaxation region in which the boundary layer could recover from the large favourable pressure gradient. Such modification of the nozzle geometry removed the vena contracta for all NPRs and significantly altered the shape and thickness of the nozzle exit boundary layer. The boundary layer's thickness and Reynolds number rapidly increased and its near-wall region returned back to a developed, turbulent...
Investigation of the development of the boundary layer within the nozzle itself for different geometries and NPRs was achieved using the low Reynolds number $k-\epsilon$ turbulence model of Launder and Sharma. The current implementation of the model was validated against low Reynolds number and relaminarising boundary layer data taken from the literature to ensure its was capable of capturing the effects of the large favourable pressure gradients and low Reynolds number boundary layers identified in the experimental measurements. Validation test cases showed the model was capable of capturing adequately both effects (in terms of global boundary layer parameters and mean velocity profile development) but also highlighted deficiencies in the ability to predict accurately turbulence statistics.

The simulations revealed the details of the role played by the favourable pressure gradient on the development of the boundary layer over a range of NPRs. The magnitude of the acceleration was shown to be sufficient for onset of relaminarisation to occur within the conical nozzles tested (based on findings reported in the current literature) and for the extent of this to increase with NPR, but acceleration was sustained over insufficient distance for the boundary layer to revert fully to a laminar state. Instead the boundary layer remained laminarescent, demonstrating some laminar-like characteristics, but was still essentially turbulent. Simulations which included a parallel nozzle exit section showed the recovery of the boundary layer to a high Reynolds number, fully turbulent state for all NPRs. The predictions also highlighted the mechanisms involved in the removal of the vena contracta, particularly the formation of a region of flow recirculation downstream of the end of the contraction and region of reduced static pressure that resulted in a region of local supersonic flow within the nozzle at high subcritical and supercritical NPRs.

A study of the influence of nozzle exit boundary layer state and shape and nozzle geometry on the development of the initial regions of high speed jet plumes was performed using LDA in order to extend current understanding. Despite order of magnitude difference in nozzle exit boundary layer thicknesses and kinematic momentum thickness Reynolds numbers, the effects of variations in the turbulent nozzle exit boundary layers on the development of the jet plume were shown to be minimal; variations in the nozzle exit boundary layer characteristics demonstrated negligible influence on the growth of the shear layer and only small differences in turbulence levels at the nozzle exit persisted for just the first 2 nozzle diameters. However, even this may have important implications for those involved in the study of jet mixing enhancement and development. The observed variations in the turbulence statistics within the mixing layer may be important to investigators who are focused on this region or whose work may be influenced by the details of nozzle exit boundary layers, such as mixing enhancement studies involving nozzle trailing edge treatments such as chevrons.
6.2 Recommendations for Further Work

Further more, at supercritical NPRs the nozzle geometry was shown to be influential in the development of the potential core region of the jet plume; variations in nozzle exit static pressure variation due to differences in nozzle geometry were shown to alter significantly the development of the shock cells which are important when studying jet plume shock noise (civil applications) and IR observability (military applications).

6.2 Recommendations for Further Work

Although validation of the Launder-Sharma low Reynolds number turbulence model showed that the model is capable of capturing the behaviour of boundary layer global parameters and mean velocity profiles during relaminarisation, it also highlighted deficiencies in the model’s ability to capture correctly boundary layer turbulence statistics, namely the excessive suppression of turbulent kinetic energy and Reynolds shear stress. Further development and refinement of the turbulence model would be beneficial, particularly if the influence of relaminarised boundary layers on jet development is to be studied using the model.

All of the experimental and numerical investigations contained within this thesis were conducted for jet flows. For the majority of practical aeronautical jet flows, total temperatures are above ambient conditions, particularly in the case of military gas turbine engine exhaust plumes. Scale tests, such as those conducted within the HPNTF at Loughborough University, are often conducted at realistic total temperatures in order to capture the effects of density ratio which would be present in the full-scale flows on the development of the jet plume. The results obtained in the present experiments have shown that for unheated conditions, the magnitude of the favourable pressure gradient was such that the nozzle exit boundary layers could be expected to start the process of relaminarisation. Increasing the total temperature will increase viscosity and velocities within the nozzle and may result in favourable pressure gradients whose magnitudes are large enough and are sustained over sufficient distance for relaminarisation perhaps to be totally completed. This may be demonstrated using a prediction of the variation of the acceleration parameter, $K$, for the LU60 nozzle at $NPR = 1.893$ (just choked) by means of isentropic 1D flow theory [62], with a range of inlet total temperature, as shown in Figure 6.1. The profiles show an increasing magnitude of the acceleration with increasing temperature, with values remaining above published critical values along the whole length of the nozzle for a total temperature of 900K. For this reason, it is recommended that the methods presented within this thesis are used to document nozzle exit conditions for varying jet total temperatures.

The addition of parallel sections was shown to remove quickly the effects of the favourable pressure gradient. However, it is not clear what length is required to remove the vena
6.2 Recommendations for Further Work

contracta and pressure gradient effects, the importance of the sudden nozzle wall slope discontinuity (recalling the smooth curve used in the Lepicovsky nozzle) or if the same effects can be recreated by simply reducing the rate of convergence toward the nozzle exit (i.e. a 'con-flare' nozzle). Further investigation into the influence of the nozzle exit geometry is certainly merited.

Finally, throughout this work there has been one question that time did not permit answering, *Is the development of the jet shear layer whose origin was a relaminarised boundary layer the same as one originating from a naturally laminar boundary layer?* Although the work of Launder\cite{75} suggested that a relaminarised boundary layer did behave in a similar fashion to a naturally laminar one, currently there is no evidence to confirm that a jet shear layer originating from a relaminarised boundary layer would behave in the same way as one originating from a naturally laminar one. In order to answer this question further work is needed. Such work would involve the design of a nozzle, or possibly a family of nozzles, contoured in such a way that an acceleration of the boundary layer is sufficient for it to relaminarise. The low Reynolds number turbulence model of Launder and Sharma, which was demonstrated to capture boundary layer relaminarisation in Chapter 3, would serve as a useful tool in the design of such nozzles, allowing concepts to be tested quickly before committing to expensive model construction and experimental testing.
Figure 6.1: Predicted Variation of the Acceleration Parameter with Total Temperature ($L_{U60}, NPR = 1.88$)
Bibliography


