The performance of compressor outlet guide vanes and downstream diffuser

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

Additional Information:

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

Metadata Record: https://dspace.lboro.ac.uk/2134/13622

Publisher: © Kim Frushard Young

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/
THE PERFORMANCE OF COMPRESSOR OUTLET GUIDE VANES
AND DOWNSTREAM DIFFUSER

by

Kim Frushard Young

A Doctoral Thesis

Submitted in partial fulfilment of the requirements for the award of
Doctor of Philosophy of the Loughborough University of Technology
16 December 1988

© by Kim Frushard Young 1988
## ERRATUM

<table>
<thead>
<tr>
<th>Page Line</th>
<th>From : To</th>
</tr>
</thead>
<tbody>
<tr>
<td>ii) 17</td>
<td>conditions : conditions</td>
</tr>
<tr>
<td>4 34</td>
<td>liklihood : likelihood</td>
</tr>
<tr>
<td>5 16</td>
<td>prescence : presence</td>
</tr>
<tr>
<td>5 33</td>
<td>corroborated : corroborated</td>
</tr>
<tr>
<td>7 9</td>
<td>is : are</td>
</tr>
<tr>
<td>12 12</td>
<td>coefficients : coefficients</td>
</tr>
<tr>
<td>12 38</td>
<td>form : from</td>
</tr>
<tr>
<td>14 4</td>
<td>practially : practically</td>
</tr>
<tr>
<td>15 11</td>
<td>devation : deviation</td>
</tr>
<tr>
<td>15 15</td>
<td>rocommend : recommend</td>
</tr>
<tr>
<td>25 2</td>
<td>infromation : information</td>
</tr>
<tr>
<td>30 9</td>
<td>Schlichting : Schlichting</td>
</tr>
<tr>
<td>37 22</td>
<td>jig : rig</td>
</tr>
<tr>
<td>51 3</td>
<td>kinetic : axial component of kinetic</td>
</tr>
<tr>
<td>51 5</td>
<td>Kinetic : Axial kinetic</td>
</tr>
<tr>
<td>51 8</td>
<td>kinetic : axial kinetic</td>
</tr>
<tr>
<td>51 14</td>
<td>dynamic : axial dynamic</td>
</tr>
<tr>
<td>51 20,21</td>
<td>delete</td>
</tr>
<tr>
<td>51 22</td>
<td>enabling the : The</td>
</tr>
<tr>
<td>52 1</td>
<td>to : can</td>
</tr>
<tr>
<td>53 32</td>
<td>form : from</td>
</tr>
<tr>
<td>114 11</td>
<td>Schlichting : Schlichting</td>
</tr>
<tr>
<td>119 12</td>
<td>form : from</td>
</tr>
</tbody>
</table>
SUMMARY

A large-scale compressor/diffuser test rig has been designed and constructed which, together with an automated data acquisition system, permits more detailed and more accurate measurements than were previously possible - especially in the region of the compressor OGV's.

Results are presented based on experiments carried out using two different single-stage axial-flow compressors operating immediately upstream of a straight-core annular diffuser, each compressor being tested with a conventional stator row and a double-dihedral chevron type of stator row i.e. four main configurations were investigated. In addition to these main tests, the effects of operating the stator row at low Reynolds number, and of operating the stator row with a small hub clearance have been investigated.

The chevron OGV shows a clear improvement, compared to the straight OGV, in terms of diffuser performance and diffuser exit velocity profiles, at the expense of a general tendency for the OGV loss to increase.

When hub clearance is used on a straight OGV, a hub corner stall arises, but this can be eliminated by incorporating blade dihedral at the hub.

It seems likely that further improvements in diffuser exit conditions could be acheived by careful design of the blade shape to encourage radial movement of flow.

Traversing within the blade passage, for a straight type OGV, has revealed significant vortices rotating in the opposite sense to classical curved duct secondary flows.
ACKNOWLEDGEMENTS

I would like to express especial thanks to Professor S.J. Stevens for his enthusiasm, encouragement and support throughout the course of this work.

My thanks also to Mr A.P. Wray, Mr R. Marson, Mr D. Glover, Mr G. Hodson, and all other members of staff from Loughborough University, Rolls-Royce and the Royal Aerospace Establishment with whom I have had dealings in connection with this research.

Finally, I would like to give praise and thanks to God for Himself, His wonderful creation, His provision and His patience in dealing with me in the years I have known Him.

This work has been supported by the Royal Aerospace Establishment, Pyestock, Farnborough under Contract/Agreement number D/ER1/9/4/2170/110/RAE(P).
LIST OF CONTENTS

Summary .................................................. ii
Acknowledgements ......................................... iii
List of Contents ........................................... iv
List of Figures .............................................. vi
List of Tables ............................................... xi
Nomenclature ................................................ xii

Chapter 1  Introduction ................................... 1
1.1  Compressor / Combustor interface .......... 2
1.2  Outlet Guide Vane; 2-d cascade work .... 4
1.3  Outlet Guide Vane; 3-d work ................. 10
   1.3.1  The Three - Dimensional Flow Environment 10
   1.3.2  Rotor Aerodynamics ......................... 11
   1.3.3  Stator Aerodynamics ....................... 15
   1.3.4  Improving Blade Performance ........... 17
   1.3.5  Wake Interaction Effects ............... 21
1.4  Annular Diffusers .................................. 24
1.5  Improving Combustor Feed .................... 29
1.6  Objectives and Scope of Present Investigation 33

Chapter 2  Experimental Facility ...................... 34
2.1  The Inlet System .................................. 35
2.2  The Working Section ............................. 37
2.3  The Exhaust System .............................. 39
2.4  Rig Configurations ................................ 40
2.5  Instrumentation .................................. 42
2.6  Traversing Mechanisms ......................... 45
2.7  Running the Rig .................................. 47

Chapter 3  Performance Evaluation and Data Analysis 49
3.1  Quantitative Performance Parameters .... 50
3.2  Graphical Techniques for Performance Assessment 56
3.3  Accuracy of Experimental Results ........ 57
3.4  Circumferential Asymmetry .................... 63
Chapter 4 Results and Discussion

4.1 'Low Stagger' Compressor Test Results
   4.1.1 Rotor Inlet Conditions
   4.1.2 Flow Conditions Between Rotor and Stator
   4.1.3 Flow conditions at OGV Exit

4.2 'High Stagger' Compressor Test Results
   4.2.1 Rotor Inlet Conditions
   4.2.2 Flow Conditions Between Rotor and Stator
   4.2.3 OGV Flow Conditions
   4.2.4 Flow Conditions at OGV Exit

4.3 'Rainbow' Compressor Test Results
   4.3.1 Proving the 'Rainbow' Technique
   4.3.2 Comparison With Tests due to Harasgama

4.4 Diffuser Test Results
   4.4.1 Diffuser Performance With Four Inlet Conditions
   4.4.2 Diffuser Reynolds Shear Stress

4.5 Overall OGV/Diffuser Performance

Chapter 5 Conclusions

Chapter 6 Recommendations for Further Work

References
Tables
Figures

Appendices
A1 Blade Design
A2 Inlet Air Filters
A3 Pressure Transducers
A4 Computer Hardware
A5 Five-Hole Probes
A6 Computer Programs
A7 Photographs
<table>
<thead>
<tr>
<th>Fig.</th>
<th>Title</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.1</td>
<td>Diffuser / Combustor arrangements</td>
<td>127</td>
</tr>
<tr>
<td>1.2</td>
<td>Schematic layout of OGV/Diffuser/Combustor</td>
<td>128</td>
</tr>
<tr>
<td>1.3</td>
<td>Cascade geometry</td>
<td>129</td>
</tr>
<tr>
<td>1.4</td>
<td>Blade surface pressure after Masek &amp; Norbury [6], - effect of separation</td>
<td>130</td>
</tr>
<tr>
<td>1.5</td>
<td>Blade surface pressure after Masek &amp; Norbury [6], - effect of incidence variation</td>
<td>131</td>
</tr>
<tr>
<td>1.6</td>
<td>Classical secondary flow patterns</td>
<td>132</td>
</tr>
<tr>
<td>1.7</td>
<td>Corner flow directions for different flow regimes</td>
<td>133</td>
</tr>
<tr>
<td>1.8</td>
<td>Rotor blade passage vortices after Inoue &amp; Kuroumaru [36]</td>
<td>134</td>
</tr>
<tr>
<td>1.9</td>
<td>Description of blade sweep and dihedral</td>
<td>135</td>
</tr>
<tr>
<td>1.10</td>
<td>Different forms of blade dihedral</td>
<td>136</td>
</tr>
<tr>
<td>1.11</td>
<td>Annular diffuser performance after Sovran &amp; Klomp [98]</td>
<td>137</td>
</tr>
<tr>
<td>1.12</td>
<td>Diffuser data after Howard et. al. [97]</td>
<td>138</td>
</tr>
<tr>
<td>2.1</td>
<td>Test rig diagram</td>
<td>139</td>
</tr>
<tr>
<td>2.2</td>
<td>Traverse station positioning</td>
<td>140</td>
</tr>
<tr>
<td>2.3</td>
<td>Probe location at Station 4 for 'low stagger' tests</td>
<td>141</td>
</tr>
<tr>
<td>2.4</td>
<td>Probe location at Station 4 for 'high stagger' tests</td>
<td>142</td>
</tr>
<tr>
<td>2.5</td>
<td>Dimensions of pressure probes</td>
<td>143</td>
</tr>
<tr>
<td>2.6</td>
<td>Interblade traversing geometry and definitions</td>
<td>144</td>
</tr>
<tr>
<td>3.1</td>
<td>Circ. distribution of axial velocity at rotor inlet (Run 1)</td>
<td>145</td>
</tr>
<tr>
<td>3.2</td>
<td>Circ. distribution of axial velocity at OGV exit (Run 1,2)</td>
<td>146</td>
</tr>
<tr>
<td>3.3</td>
<td>Circ. distribution of axial velocity at diffuser exit (Run 1,2)</td>
<td>148</td>
</tr>
<tr>
<td>3.4</td>
<td>Circ. distribution of mass flow error (Run 1,2)</td>
<td>150</td>
</tr>
<tr>
<td>3.5</td>
<td>Circ. distribution of axial velocity at rotor inlet (Run 3)</td>
<td>152</td>
</tr>
<tr>
<td>3.6</td>
<td>Circ. distribution of axial velocity at OGV exit (Run 3,4)</td>
<td>153</td>
</tr>
<tr>
<td>3.7</td>
<td>Circ. distribution of axial velocity at diffuser exit (Run 3,4)</td>
<td>155</td>
</tr>
<tr>
<td>3.8</td>
<td>Circ. distribution of mass flow error (Run 3,4)</td>
<td>157</td>
</tr>
<tr>
<td>Section</td>
<td>Description</td>
<td>Page</td>
</tr>
<tr>
<td>---------</td>
<td>-----------------------------------------------------------------------------</td>
<td>------</td>
</tr>
<tr>
<td>4.1.1</td>
<td>Axial velocity profile at rotor inlet (Run 1)</td>
<td>159</td>
</tr>
<tr>
<td>4.1.2</td>
<td>Rotor incidence (Run 1)</td>
<td>160</td>
</tr>
<tr>
<td>4.1.3</td>
<td>Axial velocity profile at rotor exit (Run 1,2)</td>
<td>161</td>
</tr>
<tr>
<td>4.1.4</td>
<td>Rotor deviation (Run 1,2)</td>
<td>162</td>
</tr>
<tr>
<td>4.1.5</td>
<td>OGV incidence (Run 1,2)</td>
<td>163</td>
</tr>
<tr>
<td>4.1.6</td>
<td>Axial velocity profile at OGV exit (Run 1,2)</td>
<td>164</td>
</tr>
<tr>
<td>4.1.7</td>
<td>Axial velocity profile at OGV exit after Harasgama [81]</td>
<td>165</td>
</tr>
<tr>
<td>4.1.8</td>
<td>Axial velocity contours at OGV exit (Run 1,2)</td>
<td>166</td>
</tr>
<tr>
<td>4.1.9</td>
<td>Velocity vectors at OGV exit (Run 1,2)</td>
<td>168</td>
</tr>
<tr>
<td>4.1.10</td>
<td>Velocity vectors at OGV exit - diffuser radial component removed,</td>
<td>170</td>
</tr>
<tr>
<td></td>
<td>(Run 1,2)</td>
<td></td>
</tr>
<tr>
<td>4.1.11</td>
<td>Yaw angles at OGV exit (Run 1,2)</td>
<td>172</td>
</tr>
<tr>
<td>4.1.12</td>
<td>OGV loss contours (Run 1,2)</td>
<td>173</td>
</tr>
<tr>
<td>4.2.1</td>
<td>Axial velocity profile at rotor inlet,</td>
<td>175</td>
</tr>
<tr>
<td></td>
<td>(Run 3,4, c.f. Howell [87], Smith [88])</td>
<td></td>
</tr>
<tr>
<td>4.2.2</td>
<td>Rotor incidence (Run 3,4)</td>
<td>177</td>
</tr>
<tr>
<td>4.2.3</td>
<td>Axial velocity profile at rotor exit (Run 3,4)</td>
<td>178</td>
</tr>
<tr>
<td>4.2.4</td>
<td>Absolute velocity profile at rotor exit (Run 3,4)</td>
<td>179</td>
</tr>
<tr>
<td>4.2.5</td>
<td>Rotor deviation (Run 3,4)</td>
<td>180</td>
</tr>
<tr>
<td>4.2.6</td>
<td>OGV incidence (Run 3,4)</td>
<td>181</td>
</tr>
<tr>
<td>4.2.7</td>
<td>Axial velocity profiles through OGV passage (Run 4)</td>
<td>182</td>
</tr>
<tr>
<td>4.2.8</td>
<td>Velocity relief maps through OGV passage (Run 4)</td>
<td>184</td>
</tr>
<tr>
<td>4.2.9</td>
<td>Axial velocity relief maps through OGV passage (Run 4)</td>
<td>185</td>
</tr>
<tr>
<td>4.2.10</td>
<td>Stagnation pressure relief maps through OGV passage (Run 4)</td>
<td>186</td>
</tr>
<tr>
<td>4.2.11</td>
<td>Static pressure relief maps through OGV passage (Run 4)</td>
<td>187</td>
</tr>
<tr>
<td>4.2.12</td>
<td>Velocity vectors through OGV passage (Run 4)</td>
<td>188</td>
</tr>
<tr>
<td>4.2.13</td>
<td>Diagram showing OGV passage vortices</td>
<td>189</td>
</tr>
<tr>
<td>4.2.14</td>
<td>Yaw angles through OGV passage (Run 4)</td>
<td>190</td>
</tr>
<tr>
<td>4.2.15</td>
<td>OGV surface pressure distribution (Run 3,4)</td>
<td>191</td>
</tr>
<tr>
<td>4.2.16</td>
<td>Blade passage pressure distribution for dihedral blade</td>
<td>194</td>
</tr>
<tr>
<td>4.2.17</td>
<td>Axial velocity profile at OGV exit (Run 3,4)</td>
<td>195</td>
</tr>
<tr>
<td>4.2.18</td>
<td>Axial velocity contours at OGV exit (Run 3,4)</td>
<td>196</td>
</tr>
<tr>
<td>4.2.19</td>
<td>Tangential blockage at OGV exit (Run 3,4)</td>
<td>198</td>
</tr>
<tr>
<td>4.2.20</td>
<td>Velocity vectors at OGV exit (Run 3,4)</td>
<td>199</td>
</tr>
</tbody>
</table>
4.2.21 Velocity vectors at OGV exit - diffuser radial component removed, (Run 3,4) 201
4.2.22 Velocity vectors at OGV exit - from inter-blade traverse (Run 4) 203
4.2.23 Yaw angles at OGV exit (Run 3,4) 204
4.2.24 OGV loss contours (Run 3,4) 205
4.3.1 Axial velocity profile at OGV exit ('rainbow' control test, Run 3) 207
4.3.2 Axial velocity contours at OGV exit ('rainbow' control test) 208
4.3.3 Axial velocity contours at OGV exit after Harasgama[81], (chevron, straight) 209
4.3.4 Axial velocity contours at OGV exit (low Reynolds number test) 211
4.3.5 Axial velocity profile at OGV exit (hub clearance test) 212
4.3.6 Axial velocity contours at OGV exit (hub clearance test) 213
4.3.7 Tangential blockage at OGV exit (hub clearance test) 214
4.4.1 Axial velocity profile at diffuser inlet (Run 1-4) 215
4.4.2 Yaw angle at diffuser inlet (Run 1-4) 216
4.4.3 Axial velocity profile at diffuser exit (Run 1-4) 217
4.4.4 Axial velocity profile down diffuser (Run 1-4) 218
4.4.5 Diffuser inner wall boundary layer parameters (Run 1-4) 222
4.4.6 Diffuser outer wall boundary layer parameters (Run 1-4) 225
4.4.7 Axial velocity contours down diffuser (Run 1) 228
4.4.8 Axial velocity contours down diffuser (Run 2) 233
4.4.9 Axial velocity contours down diffuser (Run 3) 238
4.4.10 Axial velocity contours down diffuser (Run 4) 243
4.4.11 Radial variation of OGV wake prominence (Run 1-4) 248
4.4.12 Axial decay of OGV wake (Run 1-4) 252
4.4.13 Axial decay of OGV wake after Harasgama [81] 256
4.4.14 Diffuser pressure recovery (Run 1-4) 258
4.4.15 Diffuser exit Reynolds shear stress after Stevens [95] 260
4.4.16 Reynolds shear stress down diffuser (Run 1,2) 261
4.4.17 Terms of streamwise momentum equation (Run 1) 263
4.4.18 Terms of streamwise momentum equation (Run 2) 267

A2.1 Filter arrangement in inlet plenum chamber 281
A4.1 Schematic layout of data acquisition and analysis hardware 286
A5.1 Five-hole probe labelling conventions 291
A7.1 The Test Rig 312
A7.2 Straight and Chevron Stator Blades 313
A7.3 Five-Hole Probe Calibration Rig 315
A7.4 Inter-Blade Five-Hole Probe and Associated Traverse Mechanism 316
A7.5 Radial Traverse Gear 317
A7.6 Circumferential Movement of OGV Ring 318
## LIST OF TABLES

<table>
<thead>
<tr>
<th>Table</th>
<th>Title</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>2.1</td>
<td>Positioning of blade surface pressure tappings</td>
<td>121</td>
</tr>
<tr>
<td>4.1</td>
<td>Wall boundary layer parameters</td>
<td>122</td>
</tr>
<tr>
<td>4.2</td>
<td>Diffuser performance figures</td>
<td>123</td>
</tr>
<tr>
<td>4.3</td>
<td>OGV performance figures</td>
<td>124</td>
</tr>
<tr>
<td>4.4</td>
<td>Combined OGV/Diffuser performance figures</td>
<td>124</td>
</tr>
<tr>
<td>4.5</td>
<td>Diffuser exit Reynolds shear stress figures</td>
<td>125</td>
</tr>
</tbody>
</table>
NOMENCLATURE

OGV  Outlet Guide Vane
A    area
AR   Area Ratio
AVR  Axial Velocity Ratio
AVDR Axial Velocity Density Ratio
B    blockage
C    aerofoil true chord
C_{ax}  aerofoil axial chord
C_{D}  drag coefficient
C_{L}  lift coefficient
C_{p}  pressure coefficient, or constant pressure specific heat
d    deviation angle
D_{eq}  equivalent diffusion factor (Eqn 1.2.1)
h    blade (passage) height
H    shape parameter
i    incidence angle
K    tangential blockage (Eqn 3.1.24)
L    diffuser axial length
m    mass flow
N    rotor speed
p    static pressure
P_{t}  stagnation pressure
q    dynamic head
r    radius
R    non-dimensional radius = (r - r_i) / (r_{o} - r_i),
    or specific gas constant for air
Re   Reynolds number
s    blade spacing
t    static temperature
T    stagnation temperature
u    axial velocity component
U    blade speed at mid span
v    radial velocity component
V    velocity
w    circumferential velocity component
W    diffuser wall length
$W, X, Y, Z$ circumferential traverse locations; see Fig. 2.2

$x$ axial distance

$\Delta x_{rs}$ axial gap between rotor trailing edge and stator leading edge

$z$ circumferential distance

$\alpha$ kinetic energy flux coefficient

$\beta$ air angle

$\beta'$ blade angle

$\gamma$ yaw angle, or ratio of specific heats

$\delta$ boundary layer thickness

$\delta^*$ displacement thickness

$\epsilon$ diffuser effectiveness

$\theta$ momentum thickness

$\lambda$ stagnation pressure loss

$\Lambda$ reaction

$\mu$ dynamic viscosity

$\nu$ kinematic viscosity

$\xi$ blade stagger angle

$\rho$ fluid density

$\sigma$ pitch angle

$\upsilon$ blade dihedral angle

$\phi$ flow coefficient

$\chi$ blade camber angle

$\psi$ diffuser wall angle

$\epsilon$ deflection angle

**Superscripts**

- sector area average

- $p$ pitchwise area average

- sector mass average

- $p$ pitchwise mass average

$\hat{}$ maximum

$'$ fluctuating component
Subscripts

c  wake centreline
i  inner wall
o  outer wall
p  pressure side
s  suction side
z  wake edge
1-2  inlet and exit angles respectively
2-12  station number (see Fig. 2.2), unless otherwise indicated
CHAPTER 1 INTRODUCTION

1.1 Compressor / Combustor Interface

1.2 Outlet Guide Vane; 2-d Cascade Work

1.3 Outlet Guide Vane; 3-d Work
   1.3.1 The Three - Dimensional Flow Environment
   1.3.2 Rotor Aerodynamics
   1.3.3 Stator Aerodynamics
   1.3.4 Improving Blade Performance
   1.3.5 Wake Interaction Effects

1.4 Annular Diffusers

1.5 Improving Combustor Feed

1.6 Objectives and Scope of Present Investigation
1.1 Compressor / Combustor Interface

In current day aero gas turbines there are certain design choices which are common to a large number of different engines. The most general commonality is the adoption of the turbofan concept, with an axial flow core. There are obviously some exceptions to this e.g. the Rolls-Royce Olympus engine in use in Concorde is not a bypass engine; helicopter engines are often radial flow (since frontal area is of a lower priority with helicopters than it is with fixed wing aircraft). A second feature common to a lot of large aero gas turbines is the use of an annular combustor in preference to tubular or tubo-annular. This is perhaps a reflection on the increasing pressure on engine manufacturers to produce highly efficient engines. In this respect the annular combustor has distinct advantages e.g. i) physically small in terms of length, weight and engine frontal area, ii) low pressure loss, iii) easy light-round (i.e. easy propagation of flame from the igniter plug to all burner locations around the combustor). These advantages mean that the annular combustor, when working properly, is distinctly superior to tubular designs. However, to get an annular combustor to work satisfactorily is often difficult and expensive since a circumferentially uniform outlet temperature profile is difficult to achieve and rig testing requires full engine mass flow at compressor delivery conditions. Nevertheless the annular combustor has achieved considerable popularity.

The details of annular combustor design have varied considerably over the years with assorted shapes and sizes. The way in which the air is delivered to the combustor is of great importance (see Section 1.5 for a more detailed description of combustor inlet flow requirements), and the diffuser obviously plays a major role in this connection. The primary function of the diffuser is to reduce the air speed in order to reduce combustor losses and provide a high static pressure to feed the combustor primary and dilution holes. The obvious way to diffuse the air is simply to increase the duct height, at compressor exit, in a uniform manner, thus producing an aerodynamically clean diffusing passage (see Fig.1.1a)). This approach has been used quite widely. However the very small annulus heights dictated by this design (particularly with high pressure ratios and high bypass ratios) bring about manufacturing difficulties due to the required accuracy, and also it was found that thermal expansion and contraction produced changes in geometry when the engine was running which led to undesirable asymmetry in the combustor exit temperature distribution.

In order to reduce this problem and to achieve greater flow stability, the dump diffuser (Fig.1.1b)) was introduced and is now seeing widespread application. The increase in flow stability arises as a result of the flow separation point being fixed, assuming that the diffusion being attempted is not greater than the optimum. This dump diffuser is shorter in length and is less sensitive to variations in compressor outlet velocity profile, although it incurs a higher pressure loss. Clearly those engine manufacturers who use dump diffusers view the higher
loss to be less significant than the improvements gained. A further advantage of a dump diffuser is that the cowl on the combustor head can be dispensed with, thus simplifying combustor design (Sotheran[11]). Because an adverse pressure gradient exists in a diffuser, great care must be taken in order that the flow diffuses in an efficient manner i.e. without separating from the walls. A review of relevant work on diffusers and the factors affecting their performance are discussed in Section 1.4.

The diffuser is sited downstream of compressor exit, the last stator row of the compressor acting as flow straighteners in order to align the flow in the axial direction. A review of relevant work on compressor blading is given in Sections 1.2,1.3. The layout of OGV's (Outlet Guide Vanes), diffuser and combustor head is shown in Fig.1.2.
1.2 Outlet Guide Vane; 2-d Cascade Work

To a certain degree, the compressor outlet guide vane row can be treated simply as a conventional compressor stator row. In the past, much design work has been based on cascade databases. A cascade consists essentially of a small number of aerofoils (typically 5-10) mounted in a rectangular wind-tunnel section, thus effectively being a nominally two-dimensional set-up. The purpose of using cascades is to acquire data in a relatively cheap and easy fashion on the fundamental characteristics of aerofoil performance for different permutations of design parameters. There are a number of limitations to the extent to which cascade information can be applied to actual engine design, these limitations being described in detail in Section 1.3. However, the cascade database still has a valuable part to play in providing initial design point calculations, and also cascades provide a good starting point for many investigations to understand basic aerodynamic phenomena before moving to the more complex environment of rotating turbomachinery. Therefore this section is devoted to describing some of the more important features of cascade testing which have especial relevance in gas turbine design.

Before moving on to discuss individual aspects of cascade testing, a convention needs to be established which describes the cascade geometry. Fig.1.3 depicts a cascade in diagrammatic form and clearly shows the various angles and dimensions pertaining to the cascade. Both rotors and stators can be modelled using cascades, although their precise functions differ; compressor rotor blades impart energy to the fluid, in the form of both kinetic and pressure energy, and the stator blades serve to recover the additional kinetic energy as pressure energy. Because a compressor operates by its very nature in adverse pressure gradients, great care must be taken to avoid flow separation, which is obviously detrimental to efficient operation. Therefore the compressor blades need to be designed in such a fashion as to prevent large-scale flow separation, and yet still achieve sufficient pressure rise for operational requirements. There are a number of design variables which are very much inter-related and therefore the design problem is not a simple one. Generally speaking, an increase in both blade camber and blade spacing will increase the likelihood of separation. Also, the angle at which the blades are staggered has a considerable influence. Thus one of the main functions of the cascade database is to impose practical limits on these basic parameters in order that efficient operation can be maintained by the prevention of large-scale separation. One such database is that produced by the N.A.C.A., and is published in several texts e.g Wilson[2, Horlock[3].

An additional way of assessing the likelihood of a cascade of aerofoils to separate is by means of the surface velocity variation, since this has a major effect on the growth of the blade surface boundary layers. This variation can be quantified by means of a Diffusion Factor, which is simply the ratio of maximum suction surface velocity to velocity at blade
exit. Dixon\cite{dixon} summarises the correlation due to Lieblein\cite{lieblein} who established a practical upper limit of this Diffusion Factor as being between 1.9 and 2.0. However, since the variation of surface velocity is not always known, an Equivalent Diffusion Factor which can be easily calculated from the blade geometry and operating conditions can be defined which Dixon gives as

\[
D_{eq} = \frac{\cos \beta_2}{\cos \beta_1} \left[ 1.12 + k(i - i_{ref})^{1.43} + 0.61 \left( \frac{S}{C} \right) \cos^2 \beta_1 (\tan \beta_1 - \tan \beta_2) \right]
\]

where \(k=0.0117\) for NACA 65-(A10) blades and \(k=0.007\) for C4 circular arc blades, \(i_{ref}\) = reference incidence defined as the mid-point of the working range. This Equivalent Diffusion Factor has the same practical upper limit as the Diffusion Factor (i.e. 1.9 to 2.0).

When separation occurs from the suction surface of a blade, the resulting large passage blockage causes a low pressure rise and also a high loss due to increased viscous mixing. Flow separation is often preceded, at slightly lower blade loadings, by the appearance of a laminar separation bubble on the suction surface. Masek & Norbury\cite{masek} have reported experiments in which they varied the flow incidence onto a set of compressor cascade aerofoils and measured the surface pressure distribution (test conditions being \(Re = 1.5 \times 10^5, Tu = 2.3\%\)). Their results (Fig.1.4) show quite clearly the presence of a laminar separation bubble evidenced by a small 'hump' in the suction surface pressure distribution, and also the effect on both pressure and suction surface pressures of a turbulent separation at the trailing edge. In the latter case the pressure gradient on the suction surface becomes virtually flat, and the gradient on the pressure surface curve is reduced so that the two curves again coincide at the trailing edge.

A further factor which has a profound influence on the behaviour of the suction surface boundary layer is the Reynolds number, which is usually calculated using the blade chord as the length factor. Roberts\cite{roberts} has reported results of cascade experiments in which the Reynolds number was systematically varied. He indicates that the reason why cascade performance is often poor at low flow speeds, is due to rapid growth of the suction surface laminar separation bubble. He reports that this laminar separation bubble is present only over a certain critical Reynolds number range, which he found to be about 0.7 - 1.8\( \times 10^5\) for his experiments. Below this critical range, complete laminar separation is in evidence whereas above this range conventional laminar-turbulent transition occurs with no separation bubble apparent. The limits of the critical Reynolds number range are affected by the free-stream turbulence level, which in Roberts tests was measured at 0.48% at inlet to the cascade. These observations of Reynolds number dependence are corroborated by Schlichting & Das\cite{schlichting}, who point out in addition that the critical Reynolds number range can be shifted according to variations in the turbulence level. In the critical range, an increase in turbulence (up to about
Introduction

3%) caused a decrease in loss and an increase in deflection due to reduced boundary layer growth. Any further increase in turbulence above about 3% causes the loss to increase due to dissipation. Cyrus[9] found that when operating a cascade below the critical Reynolds number (Re = 1.35x10^5 in his case, for 2-3% turbulence), extensive suction surface separation takes place, and the overall loss is increased.

In Masek & Norbury's[6] tests it is interesting to note the changes to the surface pressure distribution as the incidence is varied (Fig.1.5), aside from the above-mentioned laminar separation effects. For their test at zero degrees incidence the pressures close to the leading edge are similar on both blade surfaces, although within just a few percent chord the pressure surface values rise sharply followed by a further shallower rise right up to the trailing edge. On the suction surface, the suction peak occurs at approximately 40% chord, with appreciable gradients either side of this peak. When operated at negative incidence (Fig.1.5), the chordwise position at which the pressure surface values show a marked increase in gradient is moved rearwards, as also is the suction peak on the suction surface. Conversely, for positive incidence conditions (Fig.1.5) then the pressures for the two surfaces diverge dramatically at the leading edge, with the suction peak on the suction surface moving forward as well. For their tests Masek & Norbury used aerofoils with a sharp leading edge and hence the pressure distributions show quite substantial differences even for quite moderate incidence changes.

Another factor which can influence cascade performance is the Mach number of the approach flow. Lieblien[5] indicates that the main effect of increasing the flow Mach number is to reduce the efficient range of operating incidence. It seems though that the performance changes due to Mach number variation are relatively small when compared to the effect of Reynolds number changes, although this comment is of course limited to subsonic flows.

The way in which cascade tunnels are constructed can affect results, and of particular importance is the degree to which the flow through the cascade is two-dimensional. Any end-wall boundary layers in the tunnel will introduce some degree of three-dimensionality to the flow field. Such a situation can be a problem if the experiments being performed are supposed to be in a purely two-dimensional environment. On the other hand, it can be put to advantage as a means of making the cascade more representative of an engine situation. However for baseline tests on aerofoils it is desirable to have a purely two-dimensional flow environment. In an attempt to achieve this some tunnel designs have employed suction slots on the end-walls immediately prior to the cascade in order to remove the end-wall boundary layer. However this does not completely eliminate the problem since there will still be some boundary layer growth on the end-wall in the flow passages themselves. This can be eliminated by incorporating porous end-walls into the tunnel to remove all the boundary layer fluid. An indication of the two-dimensionality of the cascade flow can be achieved by calculation of the axial velocity ratio (AVR), which compares the free-stream axial velocity
upstream of the cascade with the average axial velocity downstream based on an integration across one blade space at midspan. Should the Mach number be sufficiently high for compressibility to be significant then the mass flow rates rather than the axial velocities are used, the parameter then being the axial velocity density ratio (AVDR).

In low speed tunnels the flow can normally be considered to be two-dimensional if the AVR is unity. If excessive end-wall suction is applied the AVR can be reduced below unity. A number of researchers have systematically varied the AVR (or AVDR) e.g. Masek & Norbury[6], Starke[10], Stark & Hoheisel[11], Gustafson[12]. General comments which can be made from these studies is that increasing the AVR causes an increase in flow deflection and a decrease in static pressure rise across the cascade. Masek & Norbury show that these trends are altered in degree by the operating incidence. They also indicate the effect of AVR on the measured loss; this was shown to be very much dependent on the flow incidence, though for zero incidence minimum loss occurred at an AVR of just greater than unity (1.03).

One result of having end-wall boundary layers present is that secondary flows are set up. Secondary flows are defined by Horlock[13] as those associated with the development of streamwise vorticity. Horlock does note however that others have chosen slightly different definitions, e.g. Dzung & Seippel[14] regard secondary flow effects as being those which disturb the two-dimensional flow structure. The predominant component of these flows takes the form of a 'squashed' vortex near each wall within the wall boundary layer, whereby the fluid immediately adjacent to the wall is swept in a pitchwise direction toward the suction surface (Fig.1.6). Gostelow[15] gives a clear explanation as to the origins of this type of flow. Essentially it arises due to the midspan pitchwise pressure gradient being imposed on the end-wall boundary layers in which the fluid is then deflected to a larger extent since it possesses lower kinetic energy. The resultant vortices can be observed downstream of the blades, and the strength of these vortices is dependent on the turning achieved within the blade row. Turbine blading is thus likely to produce stronger secondary flows than compressor blading (Bario et al.[16], Horlock[13]), and in fact this has been known as an experimental fact for some time. Secondary flows cause two problems in turbine blading; firstly the losses are significantly increased - in this connection Moore & Adhye[17] indicate that about a third of the blade losses occur downstream of the turbine blades which they attribute largely to dissipation of secondary flow vortices; and secondly if the relatively cool boundary layer fluid is exchanged for hot mainstream fluid significant cooling problems can arise. Denton & Cumpsty[18] and Walsh & Gregory-Smith[19] indicate that further encouragement to the development of secondary flows in turbines is provided by a skewed inlet boundary layer. A skewed boundary layer is one in which there is radial variation of both axial and circumferential components of velocity.

Whilst the problem of increased loss due to secondary flows is reduced for compressor blading, when compared to turbine blading, nevertheless it is still an aspect of research which
Introduction

attracts a fair amount of attention, and has done for some years. Armstrong\textsuperscript{[20]} found that secondary flows can cause local separation of the suction surface boundary layer adjacent to the end-wall. Bario et. al.\textsuperscript{[16]} also found the suction surface near the end-wall to be the region where low energy and low velocity fluid accumulated, and show how the apparent centre of the passage vortex moves toward the suction surface as well. They suggest that this corner effect is generated by interaction of the blade suction side boundary layer with the mass transport associated with the passage vortex. Marchal &Sieverding\textsuperscript{[21]} indicate that simple interaction of the two boundary layers has negligible effect in compressor cascades.

Renken\textsuperscript{[22]} suggests that the corner effect produced by the two boundary layers is of considerable significance and this contribution to the secondary flow could be easily reduced by introducing a fillet radius into the suction surface end-wall corner. However, in discussion of this point at the conference at which Renken's paper was presented, Barry pointed out that tests had been performed with such a fillet radius and rather than reduce secondary flow a large concentrated core of secondary vorticity was observed in the main stream. In an attempt to predict the effect of the use of fillets Debruge\textsuperscript{[23]} indicates that the probability of a corner separation is thereby reduced. In an investigation of corner flows in a simple 90 degree corner Zamir &Young\textsuperscript{[24]} show that a laminar corner flow will separate readily under the influence of an adverse pressure gradient. A further point of interest which Zamir & Young report is that the direction of fluid rotation in the corner flow changes in sign when the flow changes from laminar to turbulent (Fig.1.7). The direction of the flow associated with turbulent flow was the same as that observed by Renken\textsuperscript{[22]}. Armstrong\textsuperscript{[20]} has also observed a suction corner separation in a compressor cascade although he attributed it to the secondary flow itself.

Tang Yan-Ping & Chen Mao-Zhang\textsuperscript{[25]} have investigated the change in cascade corner flow produced by incorporating a strake projecting from the leading edge for a short distance upstream. The main advantage they found seems to be that high energy fluid is retained close to the corner and thus corner separation is inhibited at high incidence. This type of blade modification bears strong resemblance to the strakes fitted to some swept-wing aircraft which produce a strong vortex over the wing suction surface to maintain lift at high angles of incidence.

It is interesting to note that Langston et. al.\textsuperscript{[26]} suggest that the passage vortex originates as a separated inlet boundary layer. Renken\textsuperscript{[22]} describes the origin of observed secondary flow patterns qualitatively by superimposing flow due to pitchwise pressure gradient onto that arising due to a turbulent corner flow. Obviously such an argument relies on the assumption of a turbulent boundary layer which in his case seems to be reasonable (Renken quotes a Reynolds number of 3.0x10\textsuperscript{5}), but surely cannot be generalised since, as noted above, some researchers have been doing work investigating laminar boundary layers.

Lacor & Hirsch\textsuperscript{[27]} show that viscous effects have little influence over the development of
streamwise vorticity in a bend, although as the fluid proceeds downstream then viscous
effects will become more apparent.

Because the spanwise extent of the secondary flows is governed by the inlet boundary
layer thickness, some people regard low aspect ratio blading as being more strongly affected
by secondary flows. However, it is more correct to say that the important factor is the ratio of
inlet boundary layer thickness to blade height rather than aspect ratio, as Marchal &
Sieverding[21] have pointed out. Marchal & Sieverding conclude that it is the presence of a
leading edge vortex leg on the pressure side of the blade which initiates the passage vortex.
However if this were the case then the other leg of the vortex, on the suction side of the blade
would surely simply cancel it out, as they rotate in the opposite sense. Rather it seems as
though the net effect of this leading edge vortex would be nil as far as initiating the passage
vortex is concerned. The correct reason why the passage vortex is set up is the imposition of
the pitchwise pressure gradient at mid-span on the low energy wall boundary layer fluid.

The strength of the secondary flows is indirectly dependent on the flow Reynolds
number. Cyrus[9] indicates that at low Reynolds numbers, below the critical range, the flow
separates from the blade suction surface, and this separation results in a reduced strength
secondary flow.
1.3 Outlet Guide Vane: 3-d Work

1.3.1 The Three-Dimensional Flow Environment

Whilst the work that is done on 2-d cascades is of great importance in understanding basic flow phenomena and processes, it cannot fully describe the actual flow in real gas turbine blading for several reasons to do with similarity (or the lack of it) in flow conditions. The flow in an actual compressor, particularly for stages situated well within the machine (i.e. embedded stages), is extremely complex (Papailiou et al.\cite{28}) with the following differences from 2-d cascades:

i) The entry velocity profile possesses considerable shear in both the axial and tangential components, which leads to considerable radial variation of incidence to both the rotors and the stators. The fact that the shear of the velocity profile extends right across the annulus also means that the passage vortices are likely to occupy a large proportion of the blade passage. The production of the secondary flow will be influenced by the radial variation of loading due to the above mentioned incidence changes. Carrick\cite{29} indicates that for turbine blading the inlet skew is in the same sense as the conventional secondary flow, and therefore the inlet skew serves to increase the strength of the secondary flow and its associated loss. Boletis et al.\cite{30} reporting on turbine research indicate that skewing of the end-wall boundary layer can increase the loss from 7% to 10%. In compressor blading the inlet skew acts in the opposite sense to the conventional secondary flow, which therefore implies that the secondary loss should be decreased. No literature has been found in which the effect of inlet skew in compressors has been addressed.

ii) Considerable radial pressure gradients may exist. These are usually of a positive sense, being set up in the swirling flow to counteract centrifugal forces. However, the sign of the pressure gradient can also be negative e.g. at exit from the compressor OGV's when a diffuser with radially outward curvature is present.

iii) Wakes with large velocity deficits, and high levels of free-stream turbulence. Whilst high levels of free-stream turbulence can be generated in a cascade situation by various means, it is rather more difficult to simulate blade wakes accurately. The severity of the influence of the blade wakes on the aerodynamics of downstream bodies will of course depend on the nature of the wake - both its thickness and the associated velocity deficit, which will in turn depend on the distance downstream of the wake generating object.

iv) Rotor tip leakage flows. Such flows are very complex of themselves and have attracted a considerable amount of attention for experimental research.

The degree of influence which the Reynolds number exerts on the fluid flow in a real engine environment is varied considerably by additional factors such as turbulence levels, the
presence of wakes, Mach number, surface roughness. Balje\cite{31} has attempted to assess the importance of low Reynolds numbers on turbomachinery efficiency, but was only able to isolate general trends because of the degree of scatter present in the data due to the above-mentioned factors. In order to properly assess the influence of Reynolds number in a given application, a substantial quantity of additional information is required. This is in effect saying that each combination of blade geometry and operating environment has its own Reynolds number regime. Bullock\cite{32} indicates how losses increase at low Reynolds numbers due to laminar separation taking place, but he also stresses the considerable influence of time-unsteady effects which lower the transition Reynolds number and tend to delay flow separation. Horlock et. al.\cite{33} have performed tests where the Reynolds number effect was investigated for compressors having different aspect ratios and found that whilst a critical Reynolds number range exists for high aspect ratio (>3:1) blading, yet for low aspect ratio (<2:1) then no such criticality exists but rather deviation correlates with change in axial velocity. They also indicate that the major Reynolds number influence in both compressors and cascades is the same i.e. low numbers tend to reduce deflection and increase losses, and therefore that the laminar separation which can be observed in cascades will also occur in compressors.

Smith\cite{34} has similarly observed that although cascade tunnels exhibit criticality, tests with rotating stages do not, although he does not make any reference to aspect ratio in this connection. He also makes the point that tests on a single rotating stage, whilst providing a more realistic environment than cascade tests, may still be unrepresentative of the performance of embedded stages. However, Klein\cite{35} indicates that turbulence levels in a compressor do not increase progressively stage-by-stage, but rather that the bulk of the increase in turbulence occurs in the first stage.

1.3.2 Rotor Aerodynamics

Because of the complexity of the flows associated with rotor blades, and the improvements in measurement techniques, a considerable amount of research effort has been devoted to this area of aerodynamics particularly over the last few years. Of especial note in this connection is the work which has been done by Lakshminarayana, with various colleagues, at Pennsylvania State University. He has reported a number of experiments in which a systematic investigation of both rotor and stator aerodynamics has been performed with sophisticated instrumentation including a 3-sensor hot-wire probe rotating with the rotor. Of particular interest to a number of researchers has been the aspect of tip clearance with its associated influence. Inoue & Kuroumaru\cite{36} have investigated flows downstream of a rotor and attempted to identify the various vortices present in the exit flow i.e. secondary flow vortex, horseshoe vortex, leakage vortex, scraping vortex (Fig.1.8). They noted a high
Introduction

magnitude of vorticity for the leakage vortex near the casing and the trailing shed vortex near the hub. The wake flow becomes symmetric as the flow proceeds downstream, having initially been asymmetric. Decay rates for the various vortices were found to vary, with the trailing shed vortex decaying quickly due to interaction with the wake and the wall boundary layer. The turbulence levels in the tip leakage flows were found to be high, as confirmed by Davino & Lakshminarayana[37] who add that this high level of turbulence promotes fast decay.

Significant radial redistribution of fluid within rotor passages has been reported by Dring et. al.[38], although they also observed a hub corner stall which will obviously create substantial blockage (thus causing redistribution). The extent to which their data can be applied is perhaps limited by the very high loading of the rotor which they tested. For the range of flow coefficients at which they tested, stall was observed varying from the above-mentioned corner stall to a full-span stall. However, a general observation which can be made from their results is that there are significant radial flows in the flow passages, in the blade boundary layers, and also in the blade wakes. Since thicker wakes will be present at the higher blade loadings it is in line with expectations that they observed greater radial flows within the wake due to centrifugal effects. They summarise by saying that even with such detailed investigations of the rotor flows it will still be some time before practical analytical tools are available to cater for these mechanisms at the design stage.

Wagner et. al.[39,40] using the same compressor as that used by Joslyn & Dring indicate that hub stall and tip leakage are more dominant over the flow field than inlet wall boundary layer thickness and secondary flow.

The work done by Lakshminarayana and his co-workers covers a substantial number of journal papers. The design and construction of their test rig, along with a full description of instrumentation hardware and techniques, are given in [41]. The instrumentation employed in their investigations includes rotating and stationary hot-wires, pressure probes such as five-hole combination probes and simple pitot probes, and blade surface pressure tappings. Their research studies included measurements of inlet guide vane and stator blade flows as well as for the rotor blades. Their rotor flow investigations have looked particularly at the tip leakage flows. Lakshminarayana et. al.[42] found that the tip clearance flow originates near quarter chord with peak values occurring near mid-chord. This leakage flow is subject to intense interaction with the wall boundary layer as it progresses through the blade passage, which leads to a very complex wall velocity profile at exit from the rotor. In the companion paper [43] to this reporting the turbulence properties, it was found that very high turbulence was encountered near the tip, with the radial component of turbulence being highest. They attribute this high radial turbulence intensity to the fluid rotation, as predicted by Lakshminarayana & Reynolds[44]. The high turbulence intensities persisted to about 8% span inwards form the casing. Lakshminarayana & Pandya[45] investigated how changing the
Introduction

rotor operating point influenced tip leakage flows. They report that off-design conditions cause the local pressure difference across the blade tip to be modified which in turn has a substantial effect on the leakage velocities. However, the most substantial leakage they found to be at design conditions generally speaking.

Bettner & Elrod\cite{46} found that several factors connected with the rotor tip region have an influence on the development of the casing boundary layer such as tip clearance, stage loading, and wall roughness. This seems to tie in with the observations of Lakshminarayana & colleagues noted above, namely that the leakage flow (the nature of which will be determined by the tip region geometries and the flow coefficient) interacts intensely with the casing boundary layer. Additional to this Bettner & Elrod found that the wall roughness affects the stall margin of the rotor.

In a separate investigation Hunter & Cumpsty\cite{47} observed that the tip clearance itself could also affect the rotor stall margin. Smith & Cumpsty\cite{48} compared the performance of a normal rotor with that of a rotor operating with a casing treatment. The casing treatment comprised axial skewed slots, rather than a simple change in wall roughness. They indicate that while a number of casing treatments can improve the stall margin, they also tend to degrade efficiency. They show that by the use of the axial skewed slot casing treatment the adverse effects of tip clearance (poor stall margin) could be minimised. For the untreated case a stall was found to initiate at the blade tip whereas for the treated case the initial stall occurred at the hub. They do note however that under light loading the casing treatment offered no discernible improvement. Whilst they were not able to propose any detailed flow mechanism which explains these observations, they did note that it is the position of a blockage rather than its magnitude which is important for the initiation of a stall, and they suggest that it is the mean flow which is important. Howell\cite{49} indicates that for tip clearances in excess of 2% of blade height, then for every additional 1% clearance a 3% loss in efficiency is incurred.

Takata & Tsukuda\cite{50} in a further evaluation of compressor casing treatments also report that improvements in stall margin are possible. They also propose a mechanism for the tip clearance flows in which momentum interchange caused by the high-speed jet (i.e. the leakage flow) plays an essential role. Boyce et. al.\cite{51} have performed tests on various casing treatments in a water flow-visualisation rig. They indicate that whilst circumferential slot type of treatment does not incur a loss penalty, axial slot treatment incurs a loss penalty of about 1 to 2 per cent but produces a greater improvement in stall margin by reducing passage blockage at the tip. In order to maximise the benefit of such a treatment they also recommend that the downstream stator tips be re-designed to incorporate the changes in deviation angle which is produced.

Clearance at the hub of a rotor can occur when variable-pitch rotor blades are used, and the pressure to reduce engine costs leads to a tendency to relax manufacturing tolerances and
operate with larger clearances (Wright\[52\]). Wright has investigated the effect of changing rotor hub clearance levels and has found that increased clearance leads to a serious degradation of performance. Therefore any rotor hub gap should be kept to the smallest which is practically possible.

Dring\[53\] has performed flow visualisation tests on the behaviour of the boundary layer of a rotor blade, in which he used an engine-representative Reynolds number (approx. $5 \times 10^5$) with two different levels of free-stream turbulence and two different flow coefficients. The rotor used in his tests is the same highly loaded machine as reported in \[38\]. The results concentrated on the mid-span boundary layer behaviour. At the design flow coefficient, transition occurred at about mid-chord and the turbulent layer remained attached right up to the trailing edge. At the lower flow coefficient the point of transition moved forward, and separation was observed at about the 70% chord location. For both flow coefficients, an increase in the level of free-stream turbulence from 0.5% to 4% produced very little change in the boundary layer behaviour, presumably due to the high Reynolds number used. Dring makes the comment that as the boundary layer shape parameter increases, the layer would become more prone to radial skewing due to centrifugal and Coriolis effects.

Pouagare et. al.\[54\] have also studied rotor blade boundary layers, though they have used quantitative measurement techniques rather than flow visualisation. In regions away from the wall the streamwise component of the boundary layer is influenced mainly by the pressure gradients in the streamwise direction. They found that the radial velocity is outward at most locations, and is at a maximum at the trailing edge region. Near the tip region the radial velocity was found to be similar on both surfaces of the blade. The profile loss at mid-span was found to be lower than estimates based on two-dimensional cascade tests, though it was higher near the tip; presumably this trend is essentially a consequence of the flow redistribution within the blade passage. They also noted a significant increase in turbulence in the tip region.

The behaviour of the rotor exit flows and particularly the blade wakes have been a further aspect studied in a considerable amount of detail by Lakshminarayana and colleagues (\[55\, 44\, 56\, 57\, 58\, 59\]). Lakshminarayana & Ravindranath\[55\] discuss various factors which affect the interaction of the rotor blade wakes with the tip region flows such as the tip leakage vortices and the end-wall boundary layer. They observe that interaction effects result in slower decay of the wake and a larger wake width. Whereas at the hub the velocity profile is primarily affected by the secondary flow vortices, near the tip it is the tip leakage flow which is dominant. The wake is often considered as comprising three regions: the trailing edge region (0 - 6% of blade chord downstream of the blade trailing edge), the near wake region (6 - 40% downstream), and the far wake region (> 40% downstream). Lakshminarayana & Reynolds\[44\] found that turbulence intensities decay very rapidly in the near wake region, the radial
component of turbulent normal stress being the highest and decays more rapidly than the other components. The analysis performed by Lakshminarayana & Reynolds showed that it was rotation that was the cause of the very high radial turbulent stress. They also found that radial shear stresses were higher than streamwise shear stresses, again in agreement with their analysis. In a rather more extensive investigation Ravindranath & Lakshminarayana[56] made measurements in the trailing edge and near wake and report results for different blade loadings for a number of radial locations across the blade passage. For higher blade loadings they found that the wake shape parameter was higher and had a slower decay rate, and also the turbulence tends toward isotropy faster. The actual level of free-stream turbulence increases as the flow proceeds through the rotor, due to centrifugal and Coriolis effects. As regards the angle of the flow leaving the rotor it was found that the flow deviation angles were typically 4 - 5° below that predicted using a cascade correlation. The relative air angles were found to be similar across the blade wake in the trailing edge region whilst the absolute flow angles differ. This has the consequence that the incidence onto the downstream stator will vary according to the axial gap between the blade rows. In this connection they recommend an axial gap of about one blade chord in order for high stator incidence to be avoided.

In further reports on the mean velocity [57] and turbulence decay [58] in the near and far wakes of the rotor Ravindranath & Lakshminarayana have found that appreciable radial velocities exist within the wake, and also that a static pressure variation exists across the wake with a relatively high static pressure within the wake. With regard to the turbulence present they again refer to very high turbulence levels in the trailing edge region with asymmetric intensity profiles which are sustained longer for higher blade loading. Once again the radial turbulence intensities were found to be the highest. Reynolds et. al.[59] note that the growth of the rotor wake is different to that for a 2-d cascade.

Hirsch & Koo[60] have also reported measurements downstream of a compressor rotor. They show that the turbulence levels increase in both the wakes and the boundary layers, and that large changes in flow angle occur in the pitchwise direction from blade to blade.

From the above it can be readily deduced that the field of rotor aerodynamics is extremely complex. A prominent disturbing factor seems to be the tip clearance and the associated flows. This tip region of flow is particularly complicated and Lakshminarayana et. al.[61] indicate that still more effort is required here.

1.3.3 Stator Aerodynamics

The amount of research effort devoted to stator aerodynamics has been rather more limited than for rotors. This is perhaps due to a supposition that stator aerodynamics could be adequately described by cascade tests, or at least more adequately than rotor aerodynamics.

One of the more extensive studies of stator aerodynamics has been that performed by
Joslyn & Dring\textsuperscript{[62]} in which they made measurements on the second stator of a two-stage compressor. The data which they present for the flow within the stator blade passages is limited to surface flow visualisation, but they do present considerable quantitative measurements of the flow at inlet to and at exit from the stator row. The tests comprise of rig runs at three different flow coefficients - high flow ($\phi = 0.55$), nominal design ($\phi = 0.51$) and near stall ($\phi = 0.45$). The compressor utilises very high blade loadings; the space/chord ratio is about twice that which cascade data indicates is required in order to prevent blade stalling. This loading is evident in the stator exit flows, which involve a hub corner stall even at the high flow coefficient of 0.55. When operating at the low flow coefficient the hub corner stall was found to have grown substantially thus creating a large blockage in the flow passage. The reason why this large stall appears at the hub corner rather than the tip corner seems to be due to the poor inlet conditions near the hub, with high incidence in this region as a result of a hub corner stall on the upstream rotor. It is doubtful if such a large hub corner stall is representative of most compressors and hence in that respect the applicability of the data is somewhat limited. Nevertheless the study does provide an interesting insight into the flow behaviour particularly with a corner stall in existence. The circumferentially-mass-averaged distributions of stagnation pressure show that near the outer wall for all three flow coefficients there is a small region of apparent negative loss due to radial redistribution of the flow as it passes through the stator passage.

Joslyn & Dring\textsuperscript{[62]} suggest that this redistribution is driven by the rotor tip leakage flow that resulted from the relatively large rotor tip clearance. In contrast to these negative loss regions, high loss is observed in the region associated with the hub corner stall, which of course is more apparent at the low flow coefficient. Also shown are the measured static pressure distributions across the annulus which also have been circumferentially-mass-averaged. In general the traverse results extrapolate well to the measured hub and tip end-wall static pressures. The traverse results are compared to a free-vortex type estimation of the passage static pressure variation based on the end-wall measurements. Generally the agreement is fairly good, although there are localised offsets from the estimation both near the outer wall (at all three flow coefficients) where the low measured pressure is due to the high loss and blockage associated with the rotor tip leakage, and for the low flow coefficient across the whole annulus (but especially near the inner wall) due to the increased loss and blockage associated with the stator hub corner stall. Joslyn & Dring suggest in their conclusions that the additive influence of corner stall on downstream aerofoil rows may be one of the more important mechanisms leading to the 'repeating stage' type of flow field, though some doubt must be cast on this view since the compressor is unrepresentatively highly loaded.

Stevens & Young\textsuperscript{[63]} have also made a fairly detailed investigation of stator flows with particular emphasis on the effect of hub clearance. However since the work reported by them
forms a part of this thesis, it will be discussed in Chapter 4 rather than at this point.

1.3.4 Improving Blade Performance

Obviously it is of prime interest to the designer to achieve the highest possible performance at the highest efficiency. Therefore many different ways of improving the performance of blading have been tried ranging from variations on the basic blading geometry to blade surface treatments and localised variation of blade angle, position of maximum thickness etc. The cascade data base provides a fairly reliable guide to choosing the optimum configuration of some of the basic parameters such as inlet and outlet angles and space/chord ratio. However the cascade data base, by its very nature, is limited in the number of parameters it can be used for. An additional aspect of design is the aspect ratio of the blading (Roberts et al. [64]). Traditionally the view seems to have been held that low aspect ratios of around unity should be avoided because it was thought that secondary flows caused higher losses since they occupied a greater proportion of the blade passage. However, manufacturers have been drawn towards these lower aspect ratios (Adkins & Smith [65]) by pressure to reduce the number of blades as a means to reduce the total number of engine components and hence also engine cost. To achieve the same flow turning from a reduced number of blades requires a longer chord in order to maintain the same space/chord ratio.

Reid & Moore [66] have performed an experimental investigation into the performance effects of blade aspect ratio, in which they used stator aspect ratios of 1.26 and 1.78. Interestingly they found that the lower aspect ratio blading was better on a number of counts - pressure ratio, efficiency, stall margin, performance over whole span, higher diffusion factor, higher incidence.

Any hub clearance employed on the stator blade is likely to have a detrimental effect on performance, and should therefore be kept to a minimum. Cumpsty [67] reports that stator hub clearance is of primary importance in affecting the thickness of the inner wall boundary layer, and similarly rotor tip clearance is important at the outer wall.

It has been known for some time that one way to achieve high aerofoil performance is to re-energise the boundary layer shortly before it would otherwise separate, by either removing the low energy fluid by suction, or by 'blowing' a jet of high energy air into the boundary layer in the streamwise direction (Schlichting [68]). A relatively simple way of employing the 'blowing' type of boundary layer improvement over an aerofoil surface is to employ a slotted type of blade or a tandem blade. The slotted blade approach is used extensively on the wings of aircraft in the form of slats and flaps to increase the low speed lift in order to reduce the take-off and landing speeds. The size of the blowing slot and the relative positions of the two aerofoil sections is critical to the effectiveness of the slot. Zhou & Squire [69] have investigated the interaction effects of the wake from the upstream aerofoil section with the
boundary layer on the downstream aerofoil section. They found that the flow immediately downstream of the upstream aerofoil was particularly complex and that the major factor influencing the interaction was the turbulence in the wake. Mikolajczak et. al.\cite{70} have experimented with slotted blades in which the slot existed in the two highly loaded blade regions (i.e. near the walls), whilst the centre part of the span was left unslotted. They found that although the negative stall margin was little changed, the positive stall margin was improved over that for the unslotted blades, thus increasing the incidence range of the blade. The effectiveness of the slot was found to be dependent on chordwise location of the slot as well as on the throat area itself. The minimum loss however, was higher than that for the unslotted blades. Further disadvantages of this type of approach to blading design improvement are that greater costs would be incurred in manufacture and that the rigidity of the blade ring would be reduced.

Wisler\cite{71} has used a four-stage compressor in order to experiment with different ways of reducing loss by modification to the blades. He reports several modifications designed to improve specific regions of flow over the blade surface e.g.

i) reducing the blade camber (or curvature) near the trailing edge at the rotor hub, whilst increasing it near the leading edge, and moving the point of maximum thickness forward in order to eliminate rotor hub separation,

ii) reducing rotor leading edge tip camber, whilst increasing it near the trailing edge, and moving the point of maximum thickness rearwards in order to unload the leading edge at the tip and reduce tip leakage flows by reducing the maximum pressure difference across the rotor blade,

iii) tailoring rotor surface boundary layers by increasing the extent of laminar flow and then controlling the turbulent diffusion over the aft portion of the blade to minimise trailing edge suction surface boundary layer parameters,

iv) compensation for rotor secondary flow and tip leakage effects, which cause unnecessary acceleration changes - by changing the aerofoil shape a more controlled acceleration could be achieved,

v) introducing more stagger on the stator blades in the end-wall regions in order to reduce the losses in these regions.

Wisler records improvements in performance for the above mentioned methods. He also comments that reductions in adverse effects of tip leakage can be attained by such methods as modified wall geometry and active tip clearance control.

Interestingly the compressor used by Joslyn & Dring\cite{62} incorporated 'end-bend' type of angle variation on the stator blades. In their tests however, as noted above, they observed a large corner stall, although it seems likely that this stall is a result of the very high loading stage rather than the end-bend.

Rolls-Royce are now incorporating end-bend\cite{72} in compressor blading of production
Introduction

engines, and claim a saving of 1% fuel consumption by the use of this blading on the RB211-535E4 engine. The physical appearance of their treated blades reveals that their approach to achieving these ends has been to increase the stagger angle of the blades near the walls by increasing both blade inlet and outlet angles. The beneficial effect of this type of blade treatment is claimed to be re-energisation of the wall boundary layers. The mechanism for this improvement is presumably due to the reduction in tip loading of the blade, and a consequent reduction of tip leakage.

End-bend blade treatment may however affect the flow over a large proportion of the blade passage, since the geometry of the flow passage is changed and therefore the pitchwise pressure gradient at various radial positions will be altered and the passage secondary flows modified. Thus the full aerodynamic significance of even a fairly small change in blade geometry could be considerable.

A further way in which the blade geometry can be varied is by the use of blade sweep or dihedral (Fig.1.9). Smith & Yeh have performed a theoretical investigation of these design types, and point out how sweep often occurs in many engines in front compressor stages and rear turbine stages where the flow direction is inclined relative to the machine centreline. In the discussion of Smith & Yeh's work, Csanady suggests that the main use of dihedral is to achieve a limited control over the axial velocity distribution by bulk movement of fluid, whereas sweep enables manipulation of the passage secondary flow in order to effect a reduction in end-wall boundary layer accumulations. Gostelow however indicates that both sweep and dihedral can be used to attain radial movement of flow, but warns that care should be taken to avoid large increases in skin friction loss and also to avoid accumulation of regions of low energy fluid.

Using a two-dimensional cascade, Breugelmans et al. have investigated the effect of varying the angle of dihedral from 0° to 35°. They accomplished this using a single set of aerofoils, and made corrections to the stagger angle, as the dihedral was altered, in order to maintain the same incidence. Their tests were carried out with two different inlet conditions; thin wall boundary layer, and artificially thickened wall boundary layer. They found that the overall loss increased for increased dihedral angle, and that the losses tended to accumulate in the suction surface corner with the acute angle. Analysis of the suction surface corner flow fields revealed that a large stall was present in the acute angled corner, whereas the main factor in the obtuse angled corner was the suppression of the secondary flow. Following up these results they then tested a curved blade with both suction surface wall angles being obtuse, i.e. with radial variation of dihedral, the idea being to reduce the losses by dispensing with the acute angled corner. The dihedral variation was accomplished by using a stacking line which was shaped as an arc of a circle, such that the dihedral angle formed at the walls was 20°. However, the expected improvements were not found, but rather a large wake and corresponding high loss was observed over the bulk of the blade span away from the walls.
Introduction

The reason suggested by Breugelmans et. al. for this rather disappointing result is that laminar separation or boundary layer bubble bursting may have been enhanced by the geometrical modifications since the Reynolds number used in this test was in the critical range \(1.4 \times 10^5\) and the inlet turbulence level was low (0.8%).

There have been several papers reporting blade dihedral experiments in turbine blading from which several points can be drawn out which are also applicable to compressor blading. Shi Jing et. al.[75], Wang Xhonqi et. al.[76] and Han Wanjin et. al.[77] all report that radial pressure gradients exist when the blades have a finite dihedral angle. Wang Xhonqi et. al.[76] confirmed Breugelmans et. al.'s[74] result that the acute wall-suction surface angle exhibits increased loss whilst the obtuse angled corner shows a reduction in loss. They point out that a curved blade, i.e. with obtuse wall-suction surface angles at both ends should give improved results as the high-loss acute angled corner would then be eliminated. Han Wanjin et. al.[77] also report an improvement in the obtuse angled corner in that a corner stall is thereby suppressed, which they attribute to the radial pressure gradient which is present.

Back in the 1950's Rolls-Royce[78] made some investigations of blades with radially varying dihedral, on an annular test rig with a downstream diffuser (both walls diverging). No Reynolds number is quoted in this work but is estimated as being approximately \(3 \times 10^5\), and therefore above the critical range. They compared a conventional straight blade with a 15° and a 25° blade, these angles being the suction surface to wall corner angles (Fig.1.10). It was found that the curved vanes acted to suppress flow breakaway in the diffuser. Because the inner and outer diffuser wall angles were not equal, a further test was carried in which the curved vanes were mounted with their outer ends radial (i.e. 0° dihedral at the outer wall). This was found to further improve the diffuser exit velocity profile, with higher velocities being measured near the walls. They also indicate that for some of the tests the curved blades actually produced a lower overall loss than the straight blades, although there was a considerable amount of scatter in the results. This reduced loss is not necessarily contradictory with the result of Breugelmans et. al. since it includes the diffuser loss. Therefore it could be possible that the blade loss is actually higher than for the straight blade but this is being outweighed by a significantly lower diffuser loss. By way of recommendation they suggested that further tests be carried out with the onset of the diffuser wall divergence beginning within the vane row.

This recommendation was subsequently acted upon and the test performed[79]. In addition to the diffusing vanes, the diffuser loading was also increased with the view that this would accentuate the benefit to be obtained from the curved vanes. Unfortunately it was found that the increase in diffuser loading had been 'overdone' and the resultant high scatter of the results tended to mask any differences. Nevertheless, a qualitative trend of improvement for the curved vanes was once again discerned, in terms of a flatter velocity distribution at diffuser exit.
Following up this early work by Rolls-Royce, Harasgama & Stevens\[80\] investigated the use of chevron-shaped vanes when used as compressor outlet guide vanes for a single stage compressor operating upstream of a diffuser. The blade chord Reynolds number was $0.47 \times 10^5$, but boundary layer trip wires were installed on the stator suction surfaces and also the freestream turbulence level at stator entry was likely to have been high since it is directly downstream of the rotor. Three different stator vane geometries (Fig.1.10) were tested:

i) Conventional stator vane; no dihedral.

ii) Chevron vane; 15° dihedral at both walls.

iii) Chevron vane; 30° dihedral at both walls.

It was found that the overall OGV/diffuser performance was little changed in quantitative terms for the three configurations. However it was found that the curved vanes produced a substantial improvement in the axial velocity profile both at exit from the OGV and from the diffuser, with a particularly dramatic improvement near the inner wall. The blade losses which were measured were fairly high, the highest value being 19% for the 30° chevron blade, and the lowest 11.2% for the 15° chevron blade (the loss for the straight blade being 17.3%). The reason for the 15° blade loss being the lowest is due to the elimination of a hub corner stall seen on the straight blades, without incurring a large wake at mid-height as for the 30° blades (and also observed by Breugelmans et. al.\[74\]). However, in terms of the diffuser exit conditions the 30° blade is the superior design with stable outlet flow and a flatter velocity profile. The conditions at diffuser exit for the 15° blade were found to be unstable with intermittent patches of flow separation from the walls. Harasgama\[81\] concludes that further investigation is required in order to ascertain the flow mechanisms occurring within the flow passages of the curved blades.

1.3.5 Wake Interaction Effects

Because it is of prime interest to the engine designer to reduce engine length in order to reduce engine weight, there is considerable pressure to reduce the axial gaps between blade rows. This in turn means that wakes from the immediately upstream blade row will have reduced time to decay and hence will be more pronounced as they impinge upon the downstream blades. This is likely to have undesirable side effects such as increased blade vibration and increased noise levels. Also, the boundary layers on the surfaces of the downstream blades will be affected by strong pressure and velocity fluctuations.

This aspect of blade aerodynamics has attracted a fair amount of research interest in recent years, especially with the availability of fast-response miniature pressure transducers of the Kulite type. About ten years back Gostelow et. al.\[82\] pointed out the need for more study of unsteady flows, and suggested the approach which might be made. To demonstrate the importance of such a study they quote figures such as a 12° variation of incidence as a rotor
Introduction

wake passes over a stator leading edge, though they do not specify what axial spacing configuration this data refers to.

Gallus et. al.\cite{83} have investigated the influence of blade row spacing, as well as that of the blade number ratio (i.e. the ratio of the number of stator blades to the number of rotor blades), on a single stage compressor. Their blades were instrumented with Kulite transducers by which they were able to measure and plot the pressure fluctuations over the blade surfaces. They also note the important point that the magnitude of wake interaction effects is dependent on the machine operating point, since at lower flow rates the suction surface boundary layer growth is increased with resultant wakes of increased severity. The shape of the wake also influences the noise production, and hence this will also vary with machine operating point.

This same point concerning the machine operating point is also taken up by Gallus et. al.\cite{84} who further indicate that the larger wakes produced by stronger throttling will give rise to increased fluctuating forces. Intense fluctuating forces are obviously undesirable as they can lead to premature component failure as has been seen in some instances of prolonged compressor surge. Gallus et. al. indicate here that increased axial clearance between blade rows will give rise to lower fluctuating forces on the downstream blades.

In an investigation of rotating stall Das & Jiang\cite{85} found that a decrease of the axial gap caused a widening of the radial extent of fluid reversal associated with the rotating stall cell. At larger axial gaps the fluid reversal seems to move towards the outer part of the annulus. They also found that, as might be expected, for increased throttling the circumferential extent of fluid reversal is increased. The onset of rotating stall is usually the first indication of compressor stall or surge. Le Bot et. al.\cite{86} found for certain distributions of downstream static pressure that flow separation from the end-walls was observed instead of a rotating stall cell.

Howell\cite{87} reports that axial gaps of the order of 1/6 to 1 blade chord length produce similar performance figures, but for axial gaps of less than 1/6 blade chord then the stage pressure and temperature rise increase, which is equivalent to an increase in the 'work done factor'. However, Howell does not suggest any mechanism by way of explanation of this trend. Smith\cite{88} reports performance improvements due to reduced axial gap, and Koch & Smith\cite{89} indicate that for small axial gaps the annulus wall boundary layer becomes thinner.

Hodson\cite{90} has investigated the boundary layer behaviour of a turbine rotor, with particular reference to unsteady effects. He suggests that the high profile loss (50% higher than cascade predictions) is due to the time-dependent transitional nature of the boundary layers. He says that the axial spacing between blade rows also exercises considerable influence on the blade boundary layer transition points.

Okiishi et. al.\cite{91} indicate that as well as the compressor rotor wake dissipation loss in stator passes, interaction of the rotor wake with the stator boundary layers causes increased
stator loss which is found in both the wake and the inter-wake flows. Okiishi\cite{57}, in discussion of a paper by Ravindranath & Lakshminarayana\cite{57} indicates that reduced axial spacing between blade rows leads to higher efficiencies.

In an investigation of compressor stator boundary layers Evans\cite{92} found, as also noted by Hodson\cite{90} for a turbine blade, that the boundary layer is highly unsteady and transitional. Large variations in the boundary layer thickness were noted with time. The need for extreme caution in applying cascade results is emphasised by the large average boundary layer growth (c.f. cascade boundary layers) due to the unsteady nature of the boundary layer development.

Pfeil et. al.\cite{93} have performed a rather more classical type of experiment on the effect of wakes on the behaviour of boundary layers. To do this they used a cascade of cylinders to generate wakes, which then impinged upon the boundary layer forming on a flat plate. They found that the spacing between the cascade and the plate leading edge was important, and that the wakes force the early transition of the plate boundary layer.

Zierke & Okiishi\cite{94} report results of an investigation into unsteadiness of total pressure in an axial compressor. Time-averaged as well as fluctuating measurements were taken. They found that rotor wakes which had interacted with stator boundary layers had a lower stagnation pressure than non-interacted wakes.
1.4 Annular Diffusers

Diffuser research is a very large field and since this investigation is primarily concerned with the aerodynamics of compressor blading and the interaction of compressor and diffuser, therefore only a short overview is given here. For a more complete review of research effort into diffusers, works such as those by Stevens[95], Harasgama[81], Japikse[96] are available. Therefore data essentially for annular diffusers only is considered here.

The essential function of the diffuser is to reduce the kinetic energy of the fluid by conversion to static pressure energy, and to do this in an efficient manner. The diffuser is thus basically a diverging duct. The amount of diffusion to be performed and the length over which this is to take place are clearly of prime importance to the designer of an aircraft gas turbine. The amount of diffusion is governed essentially by the diffuser area ratio, and therefore the objective of the designer is to keep the diffuser length as short as possible whilst maintaining the required area ratio. As the length is reduced the divergence angle increases, and losses due to skin friction are outweighed by an increase in losses due to increased dissipation in the thicker wall boundary layers and eventually flow separation from the walls.

The different flow regimes within a diffuser can be described as follows:

i) No appreciable stall, with moderate diffusion rate, and the flow well behaved and unseparated.

ii) Large transitory stall in which a separation is present but its size and position is variant.

iii) Fully developed stall. In this case the high diffusion rate attempted causes the flow to separate from one wall and remain separated whilst the flow along the other wall remains attached. The region of separated fluid forms a recirculation zone.

iv) Jet flow. The divergence angle in this case is so great that the flow separates from both walls. The diffuser is thus achieving very little diffusion and the loss associated with the separation is high.

The amount of diffusion performed in a diffuser, assuming no separation, can be indicated approximately by the area ratio. In the case of an annular diffuser the geometry can be completely defined (Howard et.al.[97]) by the following parameters; the two wall angles, the inlet radii and a non-dimensional length (usually the mean wall length). The area ratio can be expressed as a function of these parameters as follows:

\[
AR = \frac{2W}{\Delta r_{inlet}} \left[ \sin \psi_o + \left( \frac{r_i}{r_o} \right)_{inlet} \sin \psi_i \right] + \left( \frac{W}{\Delta r_{inlet}} \right)^2 \left( 1 - \frac{r_i}{r_o} \right)_{inlet} \left[ \frac{\sin^2 \psi_o - \sin^2 \psi_i}{(1 + \frac{r_i}{r_o})_{inlet}} \right]
\]
Introduction

This does not however provide any information on the likelihood of the flow to separate from the walls. Such additional information can be derived from correlations of diffuser test data such as that by Sovran \& Klomp (Fig.1.11), which is also reported in other texts such as Japikse. This correlation shows that there are two different optimum design lines depending on whether the length ($C_p^*$ line) or the area ratio ($C_p^{**}$ line) is the more critical. Generally speaking the aero engine designer is more interested in the $C_p^*$ line of Sovran \& Klomp's data since engine length is a prime consideration.

The Sovran \& Klomp chart is for low inlet blockage. A near-fully developed inlet velocity profile was used by Howard et. al. (Fig. 1.12), who showed that optimum characteristics of annular diffusers are close to those of two-dimensional diffusers though for the straight-core type of annular diffuser the optimum characteristics are better. However it was also found that the margin from first transitory stall to first fixed stall was significantly smaller than for two-dimensional diffusers. The reason proposed for this was that the fully developed stall in the annular diffuser tends to form only on a small portion of the diffuser wall with the flow redistributing itself in the annulus, and therefore stall may exist in annular diffusers under less severe geometric conditions than in two-dimensional diffusers.

Usage of the charts produced by Sovran \& Klomp and Reneau et. al. will provide most of the necessary information to design a diffuser with stable (unstalled) outlet flow and a pressure recovery close to optimum. However, in practice the selection of the diffuser is not as simple as perhaps might be thought from the above, since there are a number of ways in which the flow inlet conditions can affect the diffuser flow and which therefore have to be taken into account. These factors include Reynolds number, Mach number, flow swirl, turbulence levels, shear in the velocity profile due to wall boundary layers, wakes from upstream bodies such as compressor blades.

Japikse has indicated that for annular diffusers the Reynolds number is a comparatively weak parameter as long as the flow is in the fully turbulent regime, and also that Mach number influence is minimal right up to nearly sonic inlet conditions.

The effect which entry swirl has on performance is very much dependent on the flow regimes present before it is introduced. If the outer wall flow is in worse shape than the inner wall flow then the introduction of flow swirl will tend to improve the outer wall flow at the expense of a worsening at the inner wall (Horlock, Elkersh et. al.). This change in the flow will vary in magnitude according to the amount of flow swirl introduced. If the inner wall flow is in poor shape prior to swirl being introduced then its presence will almost certainly cause flow separation from the inner wall. Essentially, what is happening when increased swirl is present is that the peak in the velocity profile is being moved radially outward i.e. a crude redistribution of the flow. Lohmann et. al. have performed a study of the effect of inlet swirl in which they report the radial movement of flow mentioned above. They also examined the effect of diffuser cant and found that for non-swirling flow then cant
Introduction

does not provide any benefit but rather causes increased losses. However, when swirling flow is present at inlet they show that the introduction of an appropriate amount of cant will increase the pressure recovery.

Because the diffuser imposes an adverse pressure gradient on the flow, the wall boundary layers tend to grow along the diffuser length. If the diffuser geometry is such that it is beyond the optimum, then this retardation of the wall boundary layer can be sufficient to cause separation of the flow at the walls and thus create a diffuser stall. Whether or not this stall is sustained and stable, depends on the degree to which the optimum geometry has been departed from. Stevens\textsuperscript{95} reports detailed results of boundary layer behaviour in three different annular diffusers having a straight core.

The effect of varying inlet conditions such as turbulence and blade wakes on the performance of annular diffusers has been studied in some depth by Stevens with various colleagues and also by Klein with others. Blockage is also a further important factor in variation of inlet conditions. The data due to Sovran & Klomp\textsuperscript{98} was acheived with a low level of inlet blockage (B=0.02) whereas that due to Howard et. al. used fully developed flow. Stevens & Williams\textsuperscript{102} performed a diffuser investigation in which the inlet blockage was varied; and found that pressure recovery decreased with increasing blockage, although with fully developed inflow the recovery again increases. Also in their investigation they found that increased turbulent mixing, caused by increased inlet turbulence, has the beneficial effect of reducing the diffuser exit kinetic energy flux coefficient without significantly affecting the diffuser total pressure loss, and therefore the pressure recovery is improved.

Japikse\textsuperscript{96} comments that the influence of inlet conditions on the performance of annular diffusers is more complicated than for conical or two-dimensional diffusers, and that complex interactions can develop within the diffuser. Thus the difficulty of correlating cause and effect precisely is increased.

A further form of distortion of the diffuser entry conditions which is encountered in turbomachinery is the presence of wakes from upstream blade rows. Usually the wakes are most prominent from the stator blades immediately upstream, but it is possible particularly if the machine is running at an off-design point for the wakes from rotor blades to be present as well.

In 1978 Stevens et. al.\textsuperscript{103} reported a diffuser investigation with wakes present from upstream blading, and included results for different inlet lengths between blade trailing edge and diffuser inlet. They found that the presence of the more severe wakes, with the blades close to diffuser inlet, did not incur any significant penalty to the diffuser performance. In fact the velocity profile at diffuser exit was found to improve rather than deteriorate.

In 1980 Klein et. al.\textsuperscript{104} presented work involving a tandem cascade at diffuser inlet, the results being seemingly contradictory to those of Stevens et. al. in that a significant increase in diffuser loss was incurred by the introduction of the blades. However, in a subsequent
paper Stevens et. al.[105] comment that it is a cause of concern that Klein et. al. report that the vane wakes were still present in the annuli surrounding the flame tube. That wakes were present in this flow region suggests that the wakes have grown rather than decayed in the diffuser; a situation which would quite clearly lead to an increased overall loss. It is worth noting that the diffuser used by Klein et. al. was highly loaded; not an optimum design. The reason why wakes should grow rather than decay in Klein's case, Stevens et. al. suggest is likely to be due to strong pressure forces in comparison to the shear forces. They also point out some of the shortcomings of the use of tandem cascades to simulate compressor exit conditions in that they do not generate the same flow unsteadiness as is associated with the flow when using an upstream rotor. It is likely that the high level of turbulence downstream of a compressor stage assists the rapid decay of blade wakes by promoting more vigorous mixing.

In their investigation devoted to the study of blade wakes on the performance of diffusers, Stevens et. al.[105] make several interesting points. They verified the result by Stevens et. al.[103] which showed that the increase in loss associated with moving the compressor OGV's close to diffuser inlet was relatively minor. They also found that the stability of the diffuser exit velocity profile was improved, this being attributed to the increased mixing due to the wakes which causes re-energisation of the diffuser wall boundary layers. Whilst moving the OGV's close to the diffuser inlet does therefore have advantages (in sufficient degree to be employed e.g. in the Rolls-Royce CE3 engine), it is also important to realise that the blade wakes do take a finite distance to decay and that for highly loaded diffusers this distance will tend to be greater. Therefore caution should be exercised to ensure that sufficient decay of the blade wakes has occurred in order for them not to be apparent in the flame tube annuli. A further point made by Stevens et. al.[105] is the necessity to take account of the diffuser presence on the performance of the blading. This factor is significant particularly when the diffuser mean flow path changes, e.g. with outwardly curved diffusers found in some current aero engines.

In a more recent paper Stevens & Wray[106] report results of an OGV/diffuser investigation in which the operating point of the compressor was varied. By this means the severity of the wakes entering the diffuser could be changed and relative wake decay rates compared. Stevens & Wray found that when operating their compressor at the off-design points, the wakes from the blades still decayed rapidly in the diffuser. For the given compressor geometry they conclude that the performance of the diffuser is relatively insensitive to the OGV outlet conditions, and does a good job of 'absorbing' the prominent wakes at the off-design points. Taking this point a little further they suggest that changes in the overall diffusion process (that is over the OGV/diffuser system) are largely determined by the performance of the OGV's. In their investigations they used OGV's with a slightly flared flow passage. This has the effect of reducing the radial pressure gradient at exit from the
OGV's, thus alleviating somewhat the high axial pressure gradient over the blade surface near the inner wall, and thereby maintaining blade performance. A further advantage of this flaring is that the pressure gradient on the diffuser outer wall is not so steep and therefore boundary layer growth is reduced somewhat.

From the various works reported above it is clear that the presence of blade wakes affects the performance of diffusers. Conversely it is also true to say that the presence of the diffuser affects the behaviour of the blade wakes. Hill et. al.\textsuperscript{107} showed how the magnitude of the axial pressure gradient affected the decay rate of wakes. If the axial pressure gradient is sufficiently large then Hill et. al. show that the wakes will grow rather than decay. Strictly speaking it is the relative magnitudes of the pressure and shear forces, rather than simply the pressure forces alone, which dictate the wake decay (or growth) rate. Therefore a deep but thin wake is much less likely to grow than is a deep and wide wake. Obviously, the greater the degree of mixing which can be promoted the better, and therefore conditions of high flow turbulence are also likely to assist in the rapid decay of wakes.
Introduction

1.5 Improving Combustor Feed

In order for the combustor to operate in a satisfactory manner there are a number of requirements which are imposed on the air-flow at entry:

i) The velocity of the air at entry to the combustor must be reasonably low in order to reduce the hot loss (i.e. the loss associated with the addition of heat to a moving airstream) to an acceptable level. Lefebvre\(^{[108]}\) indicates that the hot loss is proportional to the square of air speed:

\[
\Delta P_{\text{hot}} = \frac{1}{2} \rho u^2 \left( \frac{T_4}{T_3} - 1 \right)
\]

where \(T_3\) = compressor delivery temperature, and \(T_4\) = combustor exit temperature.

As indicated earlier in section 1.1, this reduction of the air speed is partially accomplished by means of a diffuser, which will obviously be of an annular configuration when an annular combustor is in use.

ii) A correct split of the mass flow between combustor head and inner and outer annuli surrounding the liner. Typically 20 - 25% of compressor exit mass flow is required to pass through the combustor head or snout, the remainder being divided (not necessarily equally) between the two annuli.

iii) The flow should be stable and free from any localised disturbances such as blade wakes from the compressor OGV's, in order to reduce the likelihood of combustor hot-spots and also of variations in combustor exit temperature profile.

Whilst there are many engines flying today which are operating reasonably well, there are still improvements which could be made at this region of the engine at combustor exit. This becomes clear when some of the conflict is revealed between what the compressor delivers and what the combustor ideally requires.

For instance, whilst at annulus mid-height the combustor requires a low mass flow fraction, it is precisely this region in which the actual mass flow fraction is highest at compressor exit due to the sheared nature of the velocity profile. The presence of the diffuser serves to add to this problem due to boundary layer retardation by the adverse pressure gradient.

The problem of blade wakes still being present in the air flow in the combustor annuli varies in severity between one engine design and another. On most engines it is not a problem of any significance since the blade wakes have decayed sufficiently. However, it has been found on some engines to be sufficiently severe to cause problems. Indeed in certain engines it has been observed for the wakes from the OGV's to grow rather than decay in highly loaded diffusers. This problem can be alleviated by ensuring that the diffusion rate attempted is below that at which wakes start to grow rather than decay, and also that the
diffuser is long enough to give the axial distance necessary for sufficient wake decay.

In order to achieve the desired mass flow split however, different OGV/diffuser design is necessitated rather than simply optimising existing designs since existing designs act in opposition to the requirement as outlined above. There are a number of potential solutions to redistributing the mass flow which involve modifications to either the OGV's or the diffuser, or even to both. For instance:

i) Applying wall suction to the diffuser wall boundary layers. This approach has been known about for some considerable time, and can produce very substantial improvements in very highly loaded diffusers. Schlicting\cite{68} indicates how in a highly divergent channel, boundary layer separation is prevented by the use of wall suction. Suction is also, of course, used on some aircraft wings in order to improve low speed performance by allowing the aerofoil to withstand higher incidences without separation of flow and consequent stalling. The air which has been bled off need not be discarded since on current day aero-engines cooling air is required by the HP turbine blading and could be supplied by bleed air from around compressor exit.

The use of boundary layer suction in diffusers can thus be used either to permit the use of high diffusion rate without separation, and/or can be used to improve the velocity profile (and therefore the mass flow distribution) at diffuser exit. The degree to which the mass flow can be redistributed by this method is fairly small, unless a large bleed fraction is employed, which in turn is only worth considering if the bleed air can be utilised elsewhere in the region. If the bleed air cannot be used then a significant performance penalty will be incurred due to the associated loss of energy from the core flow. It would however be capable of opposing the natural tendency of the diffuser to make the velocity profile of the compressor worse.

ii) The use of vortex generators. Again, this is really a boundary layer control type of solution. The principle is that small vortex-generating elements are positioned on the passage wall, in order to set up vortices which act to transfer momentum between the slow-moving boundary layer and the fast-moving free-stream. Whilst the usual aim of this type of modification is to prevent or delay the onset of separation, nevertheless it could still be used as a mechanism for improving the diffuser exit velocity profile and hence also the mass flow distribution.

iii) A third approach along the lines of boundary layer control is to use a VCD (Vortex Controlled Diffuser) which is a relatively recent proposition which is claimed to improve the exit velocity profile. Adkins et. al.\cite{109} describe such an installation, where differing amounts of bleed air and geometry were investigated. In essence the principle of the VCD is that a bound vortex is set up which guides the flow smoothly over a sudden expansion. For highest performance several percent of the upstream mass flow needs to be bled off from the chamber in which the vortex is retained. However by placing an almost conventional wide-
angle diffuser downstream of the vortex control slot substantial improvements can still be acheived without any air being bled off. Adkins et. al. suggest that the use of such a hybrid diffuser could lead to either an improved diffuser pressure recovery or to a shorter length.

Once more the method is useful as a way of improving the diffuser exit velocity profile. However the VCD and the hybrid diffuser are still at an early stage of development and little information exists on the total pressure loss of such a set-up.

iv) The use of splitter vanes. In an annular diffuser, these vanes would take the form of layers taken from a conical surface. Their function is to direct fluid toward the walls of the diffuser thus delaying flow separation from the walls (Chang[110]). In their experiments Cochran & Kline[111] found that, apart from permitting the use of significantly larger diffuser divergence angles, the outlet velocity profile was improved by making it more uniform. At first sight then it would seem that splitter vanes would be a good solution to the problems of acheiving the correct mass flow split to the combustor.

However, there are two disadvantages. Firstly, there will be an additional loss incurred as a result of skin friction and also the wakes from the vanes. Secondly, there is a similar problem to that experienced by faired diffusers i.e. the difficulty in maintaining geometric accuracy due to manufacturing tolerances and thermal distortions.

v) Modifications to blade shape of the compressor outlet guide vanes, in order to acheive radial movement of the fluid toward the walls. The most obvious way, perhaps, to attempt flow redistribution by this means would be to significantly increase the blade thickness near the centre of the passage. This would literally force fluid toward the walls in order to pass around the restriction. However simply introducing a restriction into the flow in this manner is very likely to cause high losses, and therefore a good deal of careful thought would be required in the design of such a modification.

Another way of changing the blade shape with a view to redistributing the flow is to employ blade sweep or dihedral, or both. Section 1.3 provides a summary of how these blade modifications can affect the fluid flow through the blade passage. The work reported on chevron blades is particularly interesting since the intended effect with this 'double-dihedral' is to move fluid toward both walls from the centre of the passage, which is precisely what is being sought after to improve air feed to the combustor. It is likely that adverse characteristics of a chevron blade would be:

i) Increased blade surface area. This implies an increase in blade loss due to increased skin friction.

ii) Corner effect on suction surface of blade at mid-height when the combined dihedral angle is high. This may perhaps culminate in a corner stall which then would progress downstream as a localised region of low energy air.

iii) Reduction of rigidity; in some engines the OGV's are used as load-bearing members to transmit mechanical loads from the hub to the casing.
Introduction

Whilst reporting the performance gains to be achieved from using chevron OGV's, Harasgama\textsuperscript{[81]} was unable to establish with any reasonable certainty the mechanism by which these improvements were brought about. Also, due to physical constraints connected with the size of his test facility Harasgama was not able to achieve accurate blade loss figures. The loss figures quoted by him were 19\% for the chevron blade compared to 17\% for conventional straight OGV's.
1.6 Objectives and Scope of Present Investigation

Since the results reported by Harasgama[81] indicated that significant gains could perhaps be made by the adoption of a chevron-shaped OGV, it was decided to follow this avenue. In his recommendations Harasgama suggested that a large-scale (relative to the size of his own test rig) test facility be manufactured which would enable easy probe access in order to accurately measure the OGV performance. This is essentially the approach which has been adopted for the initial tests reported here, though the annulus height (51.82 mm) is not as large as Harasgama recommended (102 - 127 mm). In order to attain the annulus heights suggested by Harasgama, would mean that, in order to keep the hub/tip radius ratio at a representative level, the diameter of the test facility would be impractically large for manufacturing purposes.

The first objective then of these tests would be to run a test with similar blading to that used by Harasgama in order to provide a more accurate measurement of the blade loss for both chevron and conventional blades. This test also acted as a good proving ground for the rig itself and for data acquisition procedures.

Having made this first set of tests to compare with and complement Harasgama's results, a second series of tests was envisaged, this time with more engine-representative blading. This would involve a fairly detailed survey of the air flow in the vicinity of the OGV's. The aim then of this second series of tests was to provide data from engine-representative blading with realistic entry conditions of detailed flow patterns including the behaviour of the wakes shed from the OGV's. Also, chevron blades would be tested again in this configuration, thus providing an additional data set on the performance of chevron blades.

Following on from these tests it was intended to experiment with refinements to blade shapes to further improve flow redistribution at diffuser exit. The precise direction of this phase in the test program would of course be influenced greatly by the results from the preceding tests. In particular, the success or otherwise of the chevron blade would clearly be of significance in the re-design especially with regard to the amount of dihedral which would be used.
CHAPTER 2 EXPERIMENTAL FACILITY

2.1 The Inlet System

2.2 The Working Section

2.3 The Exhaust System

2.4 Rig Configurations

2.5 Instrumentation

2.6 Traversing Mechanisms

2.7 Running the Rig
2.1 The Inlet System

The rig is of vertical construction and therefore the inlet system is situated above the working section of the test rig. Since the mass flow rates through the rig are fairly high (up to \(10 \text{ kg s}^{-1}\)), in order to reduce inlet losses to a minimum large cross-sectional area ducting is used. Again because of the high mass flow requirement, the air was drawn from outside the building. A large inlet duct (approx. 1.2m x 1.2m x 5m length) incorporating straightening vanes feeds the inlet air into a large plenum chamber (31m\(^3\)). This plenum chamber houses a bank of high performance air filters close to the inlet end of the chamber. The filters are arranged in units and since they only operate over a fairly narrow band of mass flow rate, the appropriate number of units are blanked off to achieve the mass flow rate required for the particular test being performed. See Appendix 3 for the filter layout and specification. Both the inlet duct and plenum chamber are designed to minimise noise transmission with thick walls covered with a layer of 50mm acoustic foam. Access to the plenum chamber is provided by a heavy hinged door and a ladder leading down into the laboratory.

From the downstream side of the filters, the air is drawn via a large bell-mouth intake into the entry length. The entry length is made up of a number of perspex casings (approx. 300mm length each) for both the inner and outer diameter of the annulus. Because of this method of construction the inner length can be shortened, if so desired, by removal of one or more of the inner casings thereby changing the length of the approach annulus prior to the working section.

The diameter and annulus height of the first section of the entry length corresponds to the dimensions at HP compressor exit of the Rolls-Royce RB211. Since both the mean diameter and the annulus height are significantly greater than this at the working section, the air must be diffused. This is accomplished by a diverging section of area ratio 2.2 and of fairly modest diffusion rate \((L/\Delta r_{\text{inlet}} = 16.2)\). The data of Howard et. al. \cite{971} shows this to be a very conservative design which is well clear of any possibility of stall. Fig.2.1 is a diagram of the test rig which portrays the above mentioned features. The inner casings can actually be removed right down to immediately downstream of this diffusing section, if so required. The inner casings are supported by eight steel-reinforced wooden struts which transfer the load onto the outer casings which are in turn supported by a heavy metal framework just below the working section. The axial location of the inner casing strut support section is usually immediately upstream of the diffusing section, but when the inner casings are reduced to a minimum then a larger diameter support section is used instead, this time immediately downstream of the diffusing section. Fig.2.1 illustrates the arrangement of the rig for the situation of minimum and maximum entry length. Downstream of the diffusing section, the annulus height is now approximately twice engine size whilst the mean diameter is now 1.18 times engine size (engine referred to is RB211). Prior to the rotor there is a short settling
length, of about 3 annulus heights, to ensure that wakes from the upstream support struts have decayed prior to the air reaching the rotor.
2.2 The Working Section

The working section of the test rig comprises essentially of the rotor, stator, and a diffuser (see photographs A7.1, A7.2). Two different designs of compressor (i.e. of both rotor and stator) were used in the tests; a 'low stagger' design and a 'high stagger' design (see Section 1.6).

The layout of these components in relation to each other and to the rest of the test rig is shown diagrammatically in Fig.2.1. The blades of both the rotor and the stator are manufactured in ABS (Acrylonitrile - Butadiene - Styrene) using an injection moulding process. Since the rotor is a relatively highly stressed item due to centrifugal loading, each rotor blade is closely checked for any faults - exterior faults by visual inspection, interior faults by accurately weighing when all 'flash' and runner had been removed. For the case of the 'low stagger' rotor blades an additional packing piece (again made from ABS) was required to maintain the correct spacing between blades and also to impart support and rigidity to the blades. In the low stagger case each rotor blade was pinned using a single 1/8" diameter pin to the aluminium rotor disc. The disc was machined with the necessary recess for the blade roots and holes for the retention pins.

Because of limitations on the capacity of the injection moulding machine, the rotor blades for the 'high stagger' blades did not possess a large root and therefore an alternative method of blade retention was devised. A new aluminium rotor disc was machined, this time in two parts, allowing a perspex ring to be held securely and onto which the rotor blades were attached by means of a metal insert and locating dowels cemented into the perspex ring. Static loading tests were performed in a test jig to simulate both centrifugal and aerodynamic loading, in order to check that realistic running load could be sustained.

A full description of blade design details for both low and high stagger blades is given in Appendix 1.

For both blade designs the rotor tip clearance was set to the smallest practically possible. Since the running tip clearance reduces as blade speed increases, due to centrifugal loading, the static tip clearance will not be representative of running conditions. The procedure adopted to arrive at a satisfactory running tip clearance was to gradually increase the static tip clearance until only a slightly perceptible rub was noticeable at the running speed, then to increase the static clearance by an amount equivalent to a satisfactory running tip clearance. Appendix 1 indicates the running tip clearances used. It should be borne in mind that these dimensions can only be approximate since changes in the temperature of the inlet air will cause differential thermal growth since different materials are being used together (in this case ABS, perspex, aluminium).

Another way in which thermal growth presented a problem was in axial growth causing the rotor disc to rub on the casings immediately upstream and downstream. This problem
arises because the datum axial location at which the inner and outer casings are 'tied' together is at the bottom of the diffuser and therefore all positions upstream of this location are subject to differential thermal growth.

The stator blades are manufactured by the same process as the rotor blades. However the method of fixing is different. Two perspex rings are machined, which form the inner and outer walls for the stator flow passage. The stator blades are located by dowels to the inner ring then cemented to both the inner and outer rings to form a rigid stator ring. The stators are thus shrouded at both hub and tip.

The stator carrier is positioned in the test rig in such a way that rotational movement of the ring is possible (see photograph A7.6 in Appendix 7). This rotational movement is constrained by a lug protruding through the outer casing, which is then anchored by a micrometer-type mechanism. By this means limited circumferential movement of the stator blades is acheived, which eases traversing requirements. An additional advantage of this design feature is that rings incorporating stator blades of different designs can be easily interchanged.

The diffuser is positioned immediately downstream of the stator row, the stator trailing edge coinciding with diffuser inlet. The design of the diffuser is very similar to the diffuser used by Harasgama in order for the 'low stagger' test configuration to be similar to his in view of the test objective outlined in Section 1.6. The diffusing section is actually fairly long \((L/\Delta r_{\text{diff\ inlet}} = 14.5)\) with an area ratio \(AR = 3.06\). From the data of Howard et. al. [97] in Fig.1.12a), this geometry falls just above the 'line of first stall'. However, it is unlikely that stall will be present in this diffusing section since the inlet flow is highly turbulent as a result of the upstream compressor, rather than the fully developed inflow on which the data of Howard et. al. is based. For test purposes diffuser exit is defined as that position at which \(L/\Delta r_{\text{diff\ inlet}}\) is equivalent to that used by Harasgama [81] (5.0). With the inner wall angle being zero, whilst the outer wall diverges at an angle of 7.0°, the test diffuser area ratio is \(AR = 1.685\). Comparing again with the data of Howard et. al., the test diffuser is shown to be quite conservative as it falls well below the 'line of first stall'.

\[38\]
2.3 The Exhaust System

The air leaving the diffuser is dumped into a large plenum chamber (approx. 12m³), from whence it discharges to atmosphere (outside the building) via a large cross-sectional area exhaust duct. Both the plenum chamber and exhaust duct are constructed with sound-attenuating walls which are lined with 50mm acoustic foam in similar fashion to the inlet system. At the junction between the exhaust plenum and the exhaust duct a throttle can be installed in order to fine-tune the flow coefficient. Large pressure changes across the throttle are not permissible since the plenum walls have not been designed to withstand internal pressure loading.

The drive motor for the compressor rotor is sited directly below the test rig. It has a maximum power rating of 45kW at 3000 rpm, and the power is fed from a thyristor controller. The thyristor drive unit monitors the motor speed, by means of a tachometer mounted on the drive-shaft, and continuously adjusts the current output in order to maintain an indicated speed to an accuracy of +/- 3 rpm. As a visual check on the tachometer accuracy, an additional digital tachometer has also been installed utilising an inductive probe which is triggered by a machined slot on the drive coupling.

Also housed in the exhaust plenum chamber is a centrifugal blower acting as an extractor to assist with 'starting' the test rig should it be required. Whilst it only has a fairly modest flow capability in comparison to the test rig flow, nevertheless it was found for the low stagger blading that it was sufficient to start the air flow. The high stagger compressor did not require starting assistance.
2.4 Rig Configurations

The following list summarises the differences in rig configuration between the various test runs.

Run 1 - 'low stagger' chevron

- Rotor: 'low stagger' blades, 650rpm
- OGV: 'low stagger' chevron blades, fully shrouded
- Inlet air filters: not fitted
- Entry length: short
- Exhaust throttle: fully open

Run 2 - 'low stagger' straight

- Rotor: 'low stagger' blades, 650rpm
- OGV: 'low stagger' straight blades, fully shrouded
- Inlet air filters: not fitted
- Entry length: short
- Exhaust throttle: fully open

Run 3 - 'high stagger' chevron

- Rotor: 'high stagger' blades, 2000rpm
- OGV: 'high stagger' chevron blades, fully shrouded
- Inlet air filters: fitted
- Entry length: long
- Exhaust throttle: set at 7"

Run 4 - 'high stagger' straight

- Rotor: 'high stagger' blades, 2000rpm
- OGV: 'high stagger' straight blades, fully shrouded
- Inlet air filters: fitted
- Entry length: long
- Exhaust throttle: set at 7"
Experimental Facility

Run 6 - low Re

<table>
<thead>
<tr>
<th>Component</th>
<th>Details</th>
</tr>
</thead>
<tbody>
<tr>
<td>Rotor</td>
<td>'high stagger' blades, 500rpm</td>
</tr>
<tr>
<td>OGV</td>
<td>'high stagger' straight blades, fully shrouded</td>
</tr>
<tr>
<td>Inlet air filters</td>
<td>fitted</td>
</tr>
<tr>
<td>Entry length</td>
<td>long</td>
</tr>
<tr>
<td>Exhaust throttle</td>
<td>set at 7&quot;</td>
</tr>
</tbody>
</table>

Run 8 - 'rainbow' control test

<table>
<thead>
<tr>
<th>Component</th>
<th>Details</th>
</tr>
</thead>
<tbody>
<tr>
<td>Rotor</td>
<td>'high stagger' blades, 2000rpm</td>
</tr>
<tr>
<td>OGV</td>
<td>'high stagger' 'rainbow' of 9 chevron blades in a ring of straight blades, fully shrouded</td>
</tr>
<tr>
<td>Inlet air filters</td>
<td>fitted</td>
</tr>
<tr>
<td>Entry length</td>
<td>long</td>
</tr>
<tr>
<td>Exhaust throttle</td>
<td>set at 7&quot;</td>
</tr>
</tbody>
</table>

Run 13 - 'rainbow' hub clearance

<table>
<thead>
<tr>
<th>Component</th>
<th>Details</th>
</tr>
</thead>
<tbody>
<tr>
<td>Rotor</td>
<td>'high stagger' blades, 2000rpm</td>
</tr>
<tr>
<td>OGV</td>
<td>'high stagger' 'rainbow' of 9 straight blades with 0.25mm hub clearance in a ring of fully shrouded straight blades</td>
</tr>
<tr>
<td>Inlet air filters</td>
<td>fitted</td>
</tr>
<tr>
<td>Entry length</td>
<td>long</td>
</tr>
<tr>
<td>Exhaust throttle</td>
<td>set at 7&quot;</td>
</tr>
</tbody>
</table>

For further details of:

i) blading geometry, see Appendix 1
ii) inlet air filters, see Appendix 2
iii) entry length, see Fig. 2.1
2.5 Instrumentation

In order to allow the use of a variety of probes of different sizes, bosses are provided at each traverse station which enable the attachment of the probe in either a motor-driven or a manual traverse gear. Each boss is accurately machined with positioning dowel-hole, and threaded metal inserts for secure attachment. The centre of the boss is drilled out to a diameter of 18\(\text{mm}\) into which inserts can be fitted, each insert being individually fitted to a particular boss such that the flow surface of the outer casing is kept flush. Each boss is provided with at least two inserts; one solid blank, and another drilled to admit a probe. The holes drilled for the probe were sized to be only just large enough to permit movement of the probe without binding, in order to reduce the leakage of air to a minimum. This type of instrumentation fitment has the additional advantage that probes can be easily installed and removed without the necessity to disturb the casings in any way.

Accompanying each instrumentation boss is an outer wall static pressure tapping. The tapping holes are of diameter 0.75mm and are provided with connectors for pressure tubing, these connectors being sealed when the tapping is not in use. Since nearly all the probes used have employed a cranked length of approximately 10\(\text{mm}\), therefore the static pressure tappings are axially displaced from the boss centreline by this same amount in order that the probe tip should be at the same axial position as the wall tapping.

In the diffuser the static pressure tappings were not placed in line axially, since previous work had indicated that readings could be adversely affected by the presence of upstream holes. Therefore the tappings were arranged in a helical type of distribution, with the angular displacement between adjacent tappings being equal to the angular spacing between the stator blades.

In planning the positioning of instrumentation it was envisaged that the bulk of the measurements would be taken at the same circumferential position, i.e. at the same side of the rig. Therefore the majority of the traverse bosses are situated on one side of the rig. However, at selected stations bosses were positioned in additional positions around the annulus in order that checks could be made on the degree of circumferential uniformity. All the traverse boss positions are indicated in Fig.2.2.

In order to make a representative measurement behind a set of rotating blades, all that is required for movement of the probe is freedom in the radial direction. However, downstream of a set of stationary blades, movement in both the radial and circumferential directions (relative to the stator blades) is required. This has been achieved by allowing limited circumferential movement to the OGV ring (approx. 4 blade spaces movement). Thus the need for a large number of circumferentially spaced traverse positions is obviated.

The bulk of the testing has been done using a miniature (tip diameter = 1.76 mm) five-hole probe. Pitot probes have been used at two stations; prior to rotor inlet, and at diffuser
Experimental Facility

exit. A cylindrical three-hole yawmeter probe has also been used for confirmatory yaw-angle measurements prior to the OGV leading edge. No turbulence measurements have been made, but rather the effort has been put into producing a detailed picture of the mean flow pattern. All three probe types used have a common probe stem diameter (3.25 mm) which is obviously an advantage from the point of view of probe interchangability.

The five-hole probe is a very versatile instrument since not only does it yield flow angles (in both pitch and yaw) but also the total and static pressures. Thus the mean velocity vector at each point is well defined. There are two modes in which a five-hole probe can be operated; nulled, and non-nulled. In the nulled mode an elaborate traverse mechanism is used which allows both pitch and yaw rotations of the probe in order to null the two pairs of side-holes. When the null point has been acheived then the centre hole is a direct indication of the flow stagnation pressure, and the static and dynamic pressures can be derived from a very simple calibration.

In the non-nulled mode of operation, the probe is introduced into the flow at a fixed and known attitude and all five hole pressures are logged. Then the flow quantities are derived by means of a sophisticated calibration. This method is thus much quicker and easier to use in the laboratory, but does require a good deal more effort in the calibration. However calibration and data analysis in such a situation does lend itself very well to computer operation, which if employed sensibly leads to a very simple and easy to use technique from the user's point of view. Further details of the theory, method of calibration and use are given in Appendix 5.

In situations where access to the measurement area is restricted then the non-nulled mode of operation becomes a necessity rather than an alternative.

Pitot probes have been used at the rotor inlet station in an attempt to acheive a more accurate datum mass flow measurement. It was felt that a pitot probe offered greater accuracy since the tip diameter is significantly smaller than for the five-hole probe and therefore readings closer to the wall could be obtained. Checks were carried out to ensure that the flow was truly axial (i.e. with no rotor pre-swirl) before deciding to use the pitot probes.

The other station at which pitot probes were used was at diffuser exit. The pitot probe was used in preference to the five-hole probe in order to reduce traversing time. This was again thought to be legitimate since there is little departure of the flow angle from the axial direction. Details of the dimensions of both the five-hole probes and the pitot probes are given in Fig. 2.5.

To assist in running the rig several thermocouples were installed. Firstly a thermocouple is positioned in the intake plenum chamber, which is used to monitor the stagnation temperature of the inlet air in order to maintain the rig running point (see Section 2.7). A further two thermocouples are embedded in the bearing supports for the main drive shaft, the readouts for which are permanently fixed to the outside of the rig. By this means any additional load on the motor could be quickly detected by a sharp rise in bearing temperature. Such a problem could
easily arise (and did!) if the inner casings 'pick up' on the rotor disc and start to rub. This sort of problem would otherwise be difficult to detect quickly because the motor control unit will maintain the motor speed accurately over the complete load range and therefore there would be no audible warning of impending distress.

In addition to making measurements upstream and downstream of the stator row, it was felt that in order to ascertain the flow mechanisms/patterns occurring within the blade passage that seeking to extrapolate downstream flow patterns to within the blade passages was rather uncertain, and it would be far better to make measurements actually within passages.

The first step towards this was to instrument the blade surfaces with surface pressure tappings. This was done for the high stagger blading, on both straight and chevron blades. The pressure tappings were distributed over a sector of six blades and were arranged at various radial and chordwise positions as indicated in Table 2.1. Because of the rig construction methods, the pressure tubing for these surface pressure tappings had to be routed down inside the inner casings and into the exhaust plenum, rather than going straight out through the outer casings.

The second step towards blade passage measurements involved inserting a probe within the blade passage. The requirement was for a probe capable of defining flow direction as well as magnitude, and that this probe should be capable of yielding data for the whole of the passage. Thus the probe type became a natural choice of a five-hole probe, since it fulfilled the requirements to define the flow vector at each point and also the analysis could be done using software which was already in existence. The traversing requirements were rather more difficult to meet since three independent directions of movement were required. The point of entry into the passage was decided as being from the downstream direction. The only other possibility would have been to go through the outer end-wall, but this was ruled out because of the nature of the rig construction. Circumferential and radial movement could be fairly easily achieved by, moving the OGV, and using a manual radial traverse mechanism respectively. The movement in the chordwise direction was permitted by constructing the probe to follow the same curvature as the stator blade camber line, and then rotating the probe in a special runner such that the probe tip follows the camberline all the way from stator trailing edge to leading edge. A photograph of this traverse mechanism and probe is given in Appendix 7. The traverse planes within the OGV passage are indicated in Fig.2.6.
2.6 Traversing Mechanisms

There are two basic forms of traversing which have been used for the tests described herein, which can be described as; i) manual traversing and ii) motor-driven traversing. Of these two methods the one which was used for the major part of the testing was the motor-driven mode. Manual traversing was used for the inter-blade five-hole probe. The form of the traverse mechanisms used is depicted in the photographs of Appendix 7.

The manual gear for performing a single radial traverse consists essentially of a unit machined in brass into which a probe can be clamped to fix its pitch/yaw orientation. The clamp device can then be moved in a translational manner by means of a micrometer type mechanism. The accuracy of positioning of the probe is limited by the free play inherent in the mechanism which amounts to approximately +/- 0.05mm.

The inter-blade five-hole probe requires a rather more complex traverse mechanism (Fig.A7.4 in Appendix 7) since it must allow movement in two directions (the third direction of movement being supplied by the circumferential movement of the stator ring - see Fig.A7.6 in Appendix 7), and also because access to the traverse region is rather limited. Radial movement is fairly easy to achieve, in a similar manner to using a conventional manual traverse. The rather more difficult to achieve movement in a chordwise direction within the blade passage has been realised by devising a system to rotate the probe such that the probe tip moves in an arc corresponding to the stator blade camber line. The movement is effected by means of control wires, routed from outside the casing, acting upon a runner which sits in a curved track. These control wires and the probe pressure tubing is fed through the outer casing by way of the existing instrumentation bosses. A primary consideration in the design of this traverse mechanism is that it should incur as low an aerodynamic blockage as possible since any substantial blockage will obviously affect the blade passage flow thus making any measurements unrealistic to a certain extent. Therefore it has been made of thin brass plate nominally aligned with the mean flow.

The motor-driven traverse mechanism used for the bulk of the measurements is operated almost wholly by computer control, the only manual intervention being in the traverse set-up procedure to initialise the computer with the two wall locations between which the probe is to traverse. The mechanism (see Fig.A7.5 in Appendix 7) consists of a stepper motor, a gearbox, and a rack and pinion drive mechanism to achieve a translational movement. The rack consists of a tube onto one side of which the gear teeth are machined and a clamping collar at one end. The probe is inserted into this tube and the collar tightened to prevent any probe rotation. The tube is then offered up to the gearbox and the guide wheels adjusted until the pinion meshes with the rack. The stepper motor is of the 48 steps/rev type, therefore one step is equivalent to an angular displacement of 7.5°. The gearing of the rotation is such that one step of the motor is translated to 0.02mm linear movement of the probe, although the
Experimental Facility

Positional accuracy is not quite so good (+/- 0.1mm) which is determined by the backlash in the gearing. The stepping action of the stepper motor is achieved by energising the four coils of the motor in the correct sequence of coil pairs. The correct coil sequencing is performed by a stepper motor driver chip. A purpose-designed electronic unit has been built housing two such chips with all the associated circuitry (including power supply) and up to two stepper motors can thus be connected simply by means of a plug-in cable. The signals required by this driver unit for motor direction and number of steps, are generated by the LSI 11/23 micro-computer. For details of the specification of the LSI 11/23 see Appendix 4.
2.7 Running the Rig

This section describes briefly the steps taken in preparation for running a test and also something of what actually happens during traversing. Also some of the logic involved in the data acquisition process is presented. The various steps are itemised here simply for ease of reading.

1) Warm-up of electrical equipment. The most important item here is the pressure transducers. The specification for the transducers recommends that a minimum warm-up time of 20 minutes be observed. The warm-up time used during testing always exceeded this, and wherever possible the transducers were left switched on all day for as many days as the testing sequence lasted.

   The rest of the electrical equipment (except the main drive motor) is switched on upon entering the laboratory first thing in the morning. Therefore all the instrumentation and traversing electrical items would all have been powered up for at least half an hour before traversing commenced.

2) Checking the zero drift of the transducers. With the pressure inputs disconnected, the analogue output from each transducer was measured using a digital voltmeter, and if necessary adjusted (by means of an offset adjustment potentiometer in the transducer circuitry) to bring the output to within 0.5mV of 0V. The amount of drift observed at this point tended to average about 1mV, although drift of up to 4mV was very occasionally noted. It is worth pointing out here that toward the end of the testing program it was discovered that the mains power connectors to the transducers were found to be very sensitive to not being pushed fully home. A 6-way Euro type mains distribution unit was used to supply power to the six units. On subsequent investigation it was found that a fairly small movement of the plugs could cause an interruption of the mains power to the units concerned, which can then cause a significant zero drift. Such movements of the plugs could conceivably occur during the various rig set-up procedures. However, this did not occur frequently, and when it did occur it was easily noticed as it caused the results to be meaningless. Therefore the problem is not thought to affect the accuracy of the results presented here, but rather it caused a delay in the testing schedule.

3) The probe is set up in the traverse mechanism. In order to achieve accurate alignment of the probe tip in the yaw direction, this operation is performed on the bench, which enables the probe inclination to be set within 1°.

4) The probe is offered up to the rig and secured in position for traversing. All the pressure tubes are connected to the appropriate transducers. The reference side of all the transducers is connected to the outer wall static pressure tapping prior to the rotor (i.e. at station W2).

5) The rig is started and the rotor run up to somewhere between 50 - 80% of design speed. This is to allow the various components of the rig to soak to an equilibrium temperature. This
Experimental Facility

usually takes about 10 - 15 minutes.
6) The computer is loaded with the data acquisition software and the program is started running. For a listing of the main program and the associated assembler subroutine, see Appendix 6. The computer is initialised with various parameters connected with the test run. The inlet air stagnation temperature is measured and the correct rotor speed is calculated to run at a constant $N\sqrt{T_{\text{inlet}}}$. This speed is then set. The ambient pressure is also logged in order to apply a correction to the measurements for deviations from ISA atmospheric pressure.
7) By means of a hand-held mouse the computer is initialised with the radial positions at which the probe just touches the inner and outer walls.
8) Automatic traversing is started. The probe is positioned at each point, under computer control, then a time period elapses during which the probe pressures settle. Probe readings are taken, the value stored in memory for each probe pressure being the average of 250 instantaneous readings taken over a five second interval.
9) At the end of each radial traverse, the data for the whole traverse is transferred to hard disc storage on a remote mini-computer (PDP 11/34). If a further traverse is to be performed (i.e. the measuring station is downstream of the stator vane ring), then the OGV ring is rotated to its new position and the next traverse started right away. At the end of each traverse the required rotor speed is also displayed, based on the average inlet air stagnation temperature measured during the preceding traverse.

During rig operation, the shaft bearing temperatures and the motor current were monitored visually in order to have the earliest possible warning of any malfunction with the rotating hardware.
CHAPTER 3 PERFORMANCE EVALUATION AND DATA ANALYSIS

3.1 Quantitative Performance Parameters

3.2 Graphical Techniques for Performance Assessment

3.3 Accuracy of Experimental Results

3.4 Circumferential Asymmetry
3.1 Quantitative Performance Parameters

In order that meaningful assessments and comparisons of the overall performance of the various measured flows can be made, a system for parameter calculation needs to be formulated. For mean values to be attained, integration over an area of the flow field will be required since the flow is three-dimensional in structure. For stations in which the flow is considered axisymmetric (e.g. prior to the stator blades), traverse data consists of a single radial traverse, so a simple radial integration is performed in this case. For the stations downstream of the stators, where circumferential non-uniformities exist, then a full area integration is performed over an area corresponding to one blade spacing. Before the main performance parameters can be defined, a system of weighting is required.

For the mean axial velocity the most suitable system is that of area-weighting:

$$\bar{u} = \int \frac{u}{A} \, dA$$

This then enables the calculation of mass flow rate

$$m = \int \rho u \, dA$$

3.1.2

to be simplified to

$$m = \rho \bar{u} A$$

3.1.3

Thus, the mean axial velocity of eqn. 3.1.1 can also be thought of as being the velocity of a flow in the same sized duct and with the same mass flow, but with a completely flat velocity profile i.e. the one-dimensional flow equivalent.

For other quantities mass-weighting has been used, for the following reasons:

i) It is considered by Livesey\cite{112} to be the basis of a rational and consistent system for the assessment of non-uniform flows.

ii) It does not violate the second law of thermodynamics.

iii) It is in common usage.

The mass-weighted mean value of a quantity $F$ is defined as:

$$F' = \int \frac{F}{m} \, dm$$

3.1.4

This equation can be more easily dealt with if expressed as:

$$F' = \int \frac{F \rho u}{\rho \bar{u} A} \, dA$$

3.1.5

and since density is considered constant
At this point it is helpful to define the kinetic energy flux coefficient (\( \alpha \)) which compares the kinetic energy of a non-uniform velocity profile with that of a uniform profile having the same mass flow.

\[
\text{Kinetic energy flux} = \frac{1}{2} \cdot \rho \cdot u^2
\]

which for a uniform profile is simply

\[
= \frac{1}{2} \cdot \rho \cdot (\bar{u})^2 \cdot A
\]

\[
= \frac{1}{2} \cdot \rho \cdot (\bar{u})^3 \cdot A
\]

For a non-uniform flow, kinetic energy flux

\[
= \int \frac{1}{2} \cdot \rho \cdot u^2 \cdot dA
\]

\[
= \frac{1}{2} \cdot \rho \cdot \int u^3 \cdot dA
\]

Thus

\[
\alpha = \frac{1}{2} \cdot \rho \cdot \int \frac{u^3}{\frac{1}{2} \cdot \rho \cdot (\bar{u})^3 \cdot A} \cdot dA
\]

\[
\Rightarrow \quad \alpha = \int \frac{u^3}{(\bar{u})^3 \cdot A} \cdot dA
\]

This coefficient can then be used to simplify the expression for mass-weighted mean dynamic head

\[
\bar{q} = \frac{1}{2} \cdot \rho \cdot \int \frac{u^3}{u \cdot A} \cdot dA
\]

Thus, substituting 3.1.11 into 3.1.12

\[
\bar{q} = \frac{1}{2} \cdot \rho \cdot \frac{\alpha \cdot (\bar{u})^3 \cdot A}{u \cdot A}
\]

\[
= \alpha \cdot \frac{1}{2} \cdot \rho \cdot (\bar{u})^2
\]

Hence the total pressure can be expressed as

\[
\bar{p}_t = \bar{p} + \bar{q}
\]

enabling the equations for the mass-weighted total pressure loss and static pressure recovery.
to be written e.g. for the OGV's as:

$$\lambda_{4,5} = \frac{p_{15} - p_{15}}{p_{14} - p_{4}}$$  \hspace{1cm} 3.1.15

$$C_{p_{4,5}} = \frac{p_{5} - p_{4}}{p_{14} - p_{4}}$$  \hspace{1cm} 3.1.16

Note that these coefficients have been non-dimensionalised with respect to the absolute upstream dynamic head rather than that based on the axial component.

For diffusers there are additional forms of pressure recovery, such as the maximum pressure recovery which occurs when there is zero loss

$$\hat{C}_{p_{5,12}} = 1 - \frac{\alpha_{12}}{\alpha_5} \left( \frac{1}{AR} \right)^2$$  \hspace{1cm} 3.1.17

and the ideal, one-dimensional pressure recovery

$$C'_{p_{5,12}} = 1 - \left( \frac{1}{AR} \right)^2$$  \hspace{1cm} 3.1.18

The degree to which the ideal pressure recovery is achieved can be indicated approximately by the diffuser effectiveness($\epsilon$)

$$\epsilon_{5-12} = \frac{C_{p_{5,12}}}{C'_{p_{5,12}}}$$  \hspace{1cm} 3.1.19

This parameter should not be confused with efficiency, since the ideal pressure recovery of eqn 3.1.18 is not necessarily the maximum pressure recovery. This apparent contradiction is explained by examination of eqn 3.1.17 which shows that the pressure recovery is dependent on the ratio of kinetic energy flux coefficient between inlet and exit of the flow passage under consideration, whereas the ideal pressure recovery assumes uniformity of both inlet and exit flows. Therefore it is possible for the effectiveness to be greater than unity without contravening the first law of thermodynamics. In a normal diffusing situation the kinetic energy flux coefficient will increase, as the wall boundary layers thicken due to the adverse pressure gradient. However in a flow such as found in an axial compressor blade passage this need not necessarily be the case since it is conceivable that the passage secondary flows could cause the wall boundary layers to decrease rather than increase.

In flows which exhibit departures from axisymmetry, such as the blade wakes often found in axial compressor diffusers, the calculation of boundary layer parameters becomes a little more complex than for two-dimensional flows. One possible solution is to calculate the parameters in one circumferential position, such as blade mid-passage, and follow this flow region down the diffuser wall. However, this procedure has two disadvantages;

i) It does not give a representative description of the flow, since it ignores some important
regions of the flow.

ii) It does not lend itself easily to calculation particularly if the flow is swirling and thus the circumferential position of the calculation changes with downstream distance.

The method used in this work is to calculate the parameters from the pitch-averaged velocity distribution, which overcomes the two disadvantages just listed. However, it is possible for this method to mask circumferentially localised growth and therefore this point should be borne in mind when considering results. The form of the equations used for calculating the boundary layer parameters is the same as for a two-dimensional boundary layer:

\[ \delta^* = \int_0^\delta (1 - \frac{u}{\bar{u}}) \, dr \]  

3.1.20

\[ \theta = \int_0^\delta \frac{u}{\bar{u}} \cdot (1 - \frac{u}{\bar{u}}) \, dr \]  

3.1.21

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

3.1.22

Harasgama, in an investigation on a single stage compressor & diffuser test rig has employed a slightly different form of these equations 3.1.20 and 3.1.21 in which he includes the radius ratio. However, since the overall radius ratio for the test configuration reported here is high (0.88), and also because it is trends rather than absolute values which are important, the simple two-dimensional calculation procedure has been retained here.

A further difficulty in the calculation of boundary layer parameters in a duct, is the problem of defining the edge of the boundary layer for integration purposes. The procedure adopted here is to define the boundary layer edge as the point of maximum velocity on the pitch-averaged velocity profile, and to integrate either side of this point to give the two boundary layer values (i.e. inner wall and outer wall values). If the point of maximum velocity is biased toward one of the walls such a definition of the edge of the boundary layer can cause one wall value to be significantly greater than the other wall value. These results initially suggest that one wall boundary layer is in much worse shape than the other, when in fact the difference could be not strictly a boundary layer related effect but simply a redistribution of mass flow resulting from some upstream flow process. Nevertheless, this problem is not considered to be serious since the pitch-averaged velocity profiles are also presented in this work and therefore provide a quick and easy check to see what degree of influence is likely to be exerted by a mass flow bias toward one wall.

Two slightly different ways are used of presenting the blockage caused in the circumferential direction by wakes form the stator blades. The simpler of the two parameters is the wake prominence, defined as
for a given radius. This quantity is calculated at a number of radii in order to give a radial
distribution of blockage. This quantity has been used by other authors e.g. Stevens
& Wray[106], Stevens et. al.[103], Harasgama[81]. The second parameter, the tangential
blockage, is defined by Joslyn & Dring[62] as

\[
K = 1 - \left( 1 - \frac{u_c}{u_z} \right)
\]

The main term on the r.h.s. of this equation represents the ratio of the circumferentially
area-averaged axial velocity to the axial velocity calculated from the circumferentially
mass-averaged total and static pressures and the circumferentially mass-averaged flow angles.
In the analysis of the test data at each station, one of the calculations performed is the
computation of the mass flow. This calculated value can then be compared to the mass flow at
the reference station (Station 2) at rotor inlet in order to give some sort of assessment of the
integrity of the test data. The data is then corrected to the standard mass flow. The correction
procedure involves the following steps;

i) Calculating the mass flow at each traverse station, and then comparing with the datum
figure, which is taken as that mass flow calculated for station W2.

ii) Apply the appropriate correction to the axial velocity at each data point.

iii) Calculate the change in dynamic head associated with this velocity change, for each data
point.

iv) Apportion this change in dynamic head between the total and static pressure at this point,
in a ratio of 1:1 (i.e. a correction of half the dynamic head change is applied to both total
and static pressure).

v) Re-calculate the mass-weighted mean quantities.

The correction ratio of 1:1 has been used on the basis of the findings of Harasgama &
Stevens[80] who had investigated the results of altering this ratio. Of course, the correction
which is applied should be such that the most likely source of error is corrected for e.g. if it
was known that the error in mass flow was caused by a too-high reading of total pressure,
then it would be sensible to apply the correction wholly to the total pressure.

However, for the tests reported here it was unclear what was the most likely cause of error
for each of the traverses performed. Therefore, since there were no clear pointers as to how
the correction should be applied, the compromise solution of using a 1:1 correction ratio was
employed. Calculations were performed on the un-corrected data in order to make some sort
of assessment of the effect of the correction procedure, and it is encouraging to note that there
Performance Evaluation and Data Analysis

seemed to be a somewhat more consistent trend in the corrected results than in the un-corrected. In any case the calculated mass flow errors were typically +/- 2% and as such were quite small and therefore the effect of any mass flow correction is correspondingly small.
3.2 Graphical Techniques for Performance Assessment

In order to further analyse localised disturbances in the flow field, and to reveal any trends, a number of ways of presenting data in graphical form have been used. These take the form of: two-dimensional graphs, contour plots where each contour represents a constant level in the quantity being displayed, flow vector plots in which the flow velocity vector at each data point is resolved into a plane which is usually perpendicular to the mean flow direction, isometric 'maps' over a flow area which provide a quick and easy-to-understand visual representation of the particular flow property.

In most instances the data is non-dimensionalised, with the non-dimensionalising parameter indicated on each graph.

For the bulk of the two-dimensional graphs, the data has been processed entirely by computer programs and usually this processing has included a spline curve fit to the existing data points, so as to give a smoothly flowing curve rather than an angular 'join-the-dots' type of line. In instances where this has been done the positions of the original data points are usually indicated by some kind of symbol.

In those positions where the data points are sufficiently dense to properly define abrupt changes in the flow quantity, then the spline curve fit produces very good results. However, if there are very few data points at a position of abrupt change then the spline curve fit can generate 'overshoot' type peaks in the curve. This point should always be borne in mind when traverse data is being considered which traverses an abrupt change, such as a wake from an upstream stator blade.
3.3 Accuracy of Experimental Results

There are three general areas within which errors can be introduced, these being: i) errors due to rig-related problems, ii) errors in measurement, iii) errors in data analysis.

i) Rig related errors.

a) The first item to be considered here concerns the manner in which environmental conditions are catered for. The rig is of an open-circuit type and will therefore be subject to changes in inlet air temperature and pressure. In order for results to be comparable between tests, the data has to be manipulated to that of standard inlet conditions. This procedure is described in Section 2.7. Essentially it consists of running the rig at a constant \( \frac{N}{\sqrt{V_{\text{inlet}}}} \) to compensate for temperature variations, and applying a simple correction factor to all probe measurements to correct for any departure from standard atmospheric pressure.

The pressure at any point in the flow relative to a datum, is proportional to a datum dynamic head since the stage pressure rise is a function of the inlet dynamic head (Cohen, et. al.\(^{113}\)). The operating range being considered here is fairly narrow and therefore can be regarded as being free from any Reynolds number effects.

\[
(p - p_{\text{datum}}) \propto q_{\text{datum}}
\]

\[
q_{\text{datum}} \propto \rho, u^2
\]

\[
\therefore (p - p_{\text{datum}}) \propto \rho, u^2
\]

Now \( \rho = \frac{p}{RT} \)

Therefore, for \( (p - p_{\text{datum}}) \) to be unaffected by changes in inlet air temperature, the change in temperature must be compensated by a similar change in \( u^2 \)

\[
\therefore \text{We require } T \propto u^2
\]

Now \( u \propto N \)

\[
\therefore T \propto N^2
\]
i.e. \[ N \propto \sqrt{T} \] 3.3.8

- if this condition is met then any changes in inlet air temperature will be compensated for and no temperature correction factor needs to be applied to the data. This technique also is in line with the way in which compressor performance is often given in non-dimensional form.

Changes in atmospheric pressure are compensated for by means of a correction factor. Now from eqn 3.3.2

\[
(p - p_{\text{datum}})_{\text{corrected}} = (p - p_{\text{datum}})_{\text{test}} \times \left( \frac{p_{\text{datum}}}{p_{\text{test}}} \right) \]

3.3.9

\[
= (p - p_{\text{datum}})_{\text{test}} \times \left( \frac{p_{\text{datum}}}{p_{\text{test}}} \right) \]

3.3.10

In these two corrections, the correction due to pressure change is considered to have negligible error involved. The temperature correction depends upon accurate setting of the rotor speed to maintain \( N/\sqrt{T} = \text{constant} \). Whilst the analogue tachometer fitted to the drive motor, which is used by the drive unit, gives an output which is stable, the actual level of speed indicated is subject to slight inaccuracies, and therefore a separate digital tachometer was fitted to enable accurate manual setting of the rotor speed.

b) In the rig inlet system a bank of air filters is fitted so as to prevent dirt accumulation or contamination of probes and pressure tappings. Since filter clogging could reduce the rig inlet pressure, the filter pressure drop was monitored throughout the test runs. It was found that there was minimal increase in filter pressure drop, due to the very large surface area and thus low dirt accumulation per unit area.

c) Another factor connected with the rig inlet system involves the siting of the rig inlet and exhaust ducts on the roof of the building. Because the two ducts point in opposite directions then the direction and strength of the wind will have an effect on pressure measurements. A strong wind blowing directly into the inlet duct will have the effect of reducing the effective aerodynamic resistance of the test rig seen by the compressor, and will result in a higher mass flow rate and a lower stage pressure rise.

In order to make a reasonably accurate assessment of the effect of ambient wind conditions, a short set of test runs was performed with a variable restriction in the exhaust duct to simulate the change in pressure due to wind between inlet and exhaust. The tests showed that an increase in back pressure equivalent to a wind speed of approximately 27mph blowing directly into the exhaust duct caused a 1.8% reduction in mass flow.
ii) Measurement inaccuracies.

These inaccuracies arise from the probe, the pressure transducers, and the computer ADC (Analogue to Digital Converter).

a) Probe related errors.

The bulk of the measurements were taken using five-hole probes. These instruments are robust, reliable and provide comprehensive information on the time-average flow structure. The probes are calibrated (see Appendix 5) using the same transducers and computer ADC as are used when testing. The calibration is performed over a wide range of flow angles - typically \( +/\pm 36^\circ \) of yaw and pseudo pitch, and comprises a large number of points so as to minimise interpolation errors. Because of the very small bore size of the probe tubes they are prone to blockage from either large particles (such as insects) or simply due to dust accumulation. For the 'low stagger' tests which were performed with unfiltered inlet air, the probe was thoroughly cleaned before each area traverse and also the data was checked for any signs of blocked probe holes. For the 'high stagger' tests, the inlet air was filtered and as a result no further problems with air-borne dust or dirt were encountered.

To check the accuracy of the probe calibration, some control experiments were performed in the calibration jig by Wray which showed that angles and velocities derived from the calibration were in extremely good agreement, and that any error involved in the calibration-and-use technique was negligible. Tarnigniaux & Oates have investigated wall proximity effects, though their probe had a conical head rather than a truncated pyramid. The maximum error in pitch angle which they noted was \( 2^\circ \), at a position of less than one probe diameter from the wall. Sitaram et. al. have also investigated wall vicinity effects and found that the error in velocity measurements amounts to approximately 1%. In the test environment encountered here, whilst solid walls are present, because the overall quantitative measurements for the major part are well away from these walls then the wall proximity effects can be treated as being negligible. The only situation where these effects might possibly become apparent is in the flow vector plots at those data points close to a solid wall.

When a five hole probe (or any multi-sensor probe for that matter) is used in a highly sheared flow then the probe readings will be inaccurate simply because of the finite distance between the measurement holes on the probe head. Sitaram et. al. judge that this spatial error is likely to be the dominant error for five-hole probes. One way to solve this problem would be to make small movements to the probe in order that all the holes could in turn be placed at exactly the same point. However, this is an extremely inefficient solution since apart from the extra traversing sophistication, it would also incur approximately a five-fold increase.
Performance Evaluation and Data Analysis

in testing time. An alternative technique (which is the one used here) is to perform the area traverse with the probe in one position only for each data point, and then to perform an interpolation in both radial and circumferential directions on the individual probe hole readings in order to collapse the pressures for all five holes onto one point. This procedure lends itself well to computer manipulation, and is described fully by Wray[117].

The effect of turbulence on five-hole probe measurements has been assessed by Sitaram et. al.[116], who indicate likely errors in velocity of only 0.33% for turbulence intensities as high as 10%. The only locations in the current test rig where the turbulence intensities are likely to be as high as 10% are immediately downstream of the blade rows where wakes and (in the case of the rotor) tip leakage flows are present. At all other stations the turbulence level is likely to be well below this value.

Sitaram et. al.[116] indicate that whilst Mach number effects can be ignored if the Mach number is less than 0.3, Reynolds number effects are of some significance. One way of avoiding these effects is to calibrate the probe at a similar air speed to that likely to be encountered during flow measurement on the test rig; which has been done in these tests.

Pitot probes have been used for a small number of the tests reported here. The stations at which they have been used are prior to the rotor, and at diffuser exit. At both these positions there is negligible flow curvature or swirl, and hence no significant problem is considered to have effect with these probes.

b) Transducer related errors.

The specification for the transducers is given in Appendix 3. The calibration of the transducers was checked, and it was found that up to about 85% of their working range the performance was very good with linearity well within the specification. Beyond this range the performance deteriorated slightly, and hence it was ensured that the measuring range fell within this 85% limit.

Before each run the transducer offset was checked and adjusted as described in Section 2.6. Because of the high working range of the transducers (+/- 500mm H2O), then any offset error becomes a larger quantity relative to the signal level. Whilst normally the transducer drift was confined to about +/- 1mV, occasionally drifts of up 4mV were observed. The likely reason for this large drift is given in Section 2.7.

In order to make an assessment of the effect of transducer drift, a raw data file from a single radial traverse between rotor and stator was run through the various analysis programs with a somewhat pessimistic value of 'simulated offset' (6mV) applied to various of the probe holes. Three aspects in particular were concentrated upon; the effect on mass flow, the effect on OGV total pressure loss, the effect on flow angles. For the worst case combinations of offset the maximum errors incurred were;
- approximately 4% mass flow error.
- a change of approximately 0.02 in the OGV total pressure loss coefficient.
- a yaw angle change of approximately 1° when the nominal yaw angle was close to zero, rising to 2° for nominal yaw angles as high as 30°.

Unfortunately no record was kept of the transducer drifts which occurred during testing, and therefore no compensation can be made to the data for any errors thus incurred. Nevertheless the above experimental results do provide an indication of the degree to which the results can be affected.

c) Analogue to Digital Converter.

The ADC is a 12-bit device, the specification for which is given in Appendix 4. Of the various types of error which the ADC is prone to, the most significant is the inherent quantising error. However this is still fairly small amounting to +/- 0.5mV. This can be further improved by taking multiple readings and then averaging. In all the tests reported here 250 individual readings were logged at each data point over a five second interval, and then averaged. Hence the error incurred in the ADC can be considered to be negligible in comparison with the errors involved in transducer drift. The calibration of the ADC was checked at several points during the test program and was found to be very good, requiring only a minimal adjustment to restore the calibration. When the transducers were first used in conjunction with the computer ADC it was found that the transducer signal output was very noisy. To solve this problem simple low-pass R-C filters (10Hz cut-off) were connected to each of the signal outputs.

iii) Data analysis problems.

a) Round-off errors are considered to be negligible since all data processing is carried out by computer which is a 16-bit machine working with floating point numbers.

b) At those stations downstream of the OGV's, the reliability of the data will obviously be dependent on the accuracy of setting the blades over that one blade passage over which the traverse has been performed since this is taken to be representative of the entire blade ring. This sort of error is difficult to quantify without sophisticated measuring equipment, but visual inspection revealed no discernible setting errors.

c) The other source of error in the manipulation of the test data is the use a cubic spline curve fitting technique. Whilst in most circumstances this curve fit produces excellent results, when a situation is encountered where the number of data points is small close to a region of abrupt change (such as a blade wake) then inaccuracies can occur with 'overshoot' of the curve occurring. However these regions are generally those in which the mass flow is
Performance Evaluation and Data Analysis

relatively low and hence the effect on the mass-weighted mean flow quantities will be small. This problem is not considered to be significant in the overall results.
3.4 Circumferential Asymmetry

In an ideal test situation the flow conditions into, through and out of the working section of the test rig should be repeatable at any point around the circumference at the same axial location. At Stations 2, 5 and 12 on the test rig, traverse facilities exist at four circumferential positions thus enabling a check of the circumferential (non-)uniformity to be made.

Fig. 3.1 shows the axial velocity profiles at Station 2 for the 'low stagger' runs and indicates excellent agreement between the four positions. The situation at diffuser inlet shows that the circumferential asymmetry is noticeably greater for both chevron and straight 'low stagger' blades. The cause of this asymmetry was traced to a deformation of the OGV carrier ring caused by the method of construction. The effect of the distortion was for local regions of the ring to move slightly in the radial direction, when aerodynamic loading was applied, such that small steps in the otherwise smooth flow surface of the casings were formed. The resulting asymmetry of the four velocity profiles is also the cause of the circumferential asymmetry seen at diffuser exit (Fig. 3.3). The computed mass flow discrepancy at the positions around the annulus are shown in Fig. 3.4, from which a consistent trend for the mass flow to be lowest at the Y position is noticeable.

For the 'high stagger' blades the problem with distortion of the OGV carrier ring was eliminated by incorporating a more rigid restraint of the ring. As a result of this modification to the test rig, the velocity profiles around the annulus for the 'high stagger' blades show good agreement at all stations along the working section (Fig's 3.5, 3.6, 3.7). The corresponding plots of computed mass flow discrepancy are given in Fig. 3.8. That the discrepancies tend to be higher than for the 'low stagger' blades is a reflection of the lower axial velocities for the 'high stagger' tests.

In summary, the circumferential uniformity throughout the rig is seen to be quite acceptable.
CHAPTER 4 RESULTS AND DISCUSSION

4.1 'Low Stagger' Compressor Test Results
   4.1.1 Rotor Inlet Conditions
   4.1.2 Flow Conditions Between Rotor and Stator
   4.1.3 Flow Conditions at OGV Exit

4.2 'High Stagger' Compressor Test Results
   4.2.1 Rotor Inlet Conditions
   4.2.2 Flow Conditions Between Rotor and Stator
   4.2.3 OGV Flow Conditions
   4.2.4 Flow Conditions at OGV Exit

4.3 'Rainbow' Compressor Test Results
   4.3.1 Proving the 'Rainbow' Technique
   4.3.2 Comparison With Tests Due to Harasgama

4.4 Diffuser Test Results
   4.4.1 Diffuser Performance With Four Inlet Conditions
   4.4.2 Diffuser Reynolds Shear Stress

4.5 Overall OGV/Diffuser Performance
4.1 'Low Stagger' Compressor Test Results

4.1.1 Rotor Inlet Conditions

With the rig having been set up in the 'low stagger' configuration described in Section 2.4, traverses were carried out to establish the flow conditions prior to the rotor. Initial traverses were performed at Station 2 with a three-hole yawmeter probe to establish that no pre-swirl was introduced by the rotor.

That the velocity profile is largely flat is a result of the short annular entry length in use for these 'low stagger' tests. The use of a longer entry length was prohibited by the inability of the compressor to overcome the additional loss. One solution to this problem would be to incorporate an auxiliary exhaust fan producing sufficient pressure rise to restore the compressor operating point back onto design. However, whilst an auxiliary exhaust fan was sited in the exhaust plenum chamber for the purpose of assisting to 'start' the test rig compressor, it was not of sufficient capacity to cope with the full-speed rig mass flow rate. A larger fan was not fitted because of the additional complexity and cost which would be incurred.

It will be noted from Fig.4.1.1 that the boundary layer attached to the inner wall is significantly thinner than that associated with the outer wall. This is a consequence of the much shorter length over which the inner wall boundary layer develops.

Since the axial velocity profile and the rotor speed are known, the radial distribution of rotor incidence can be calculated and is shown in Fig.4.1.2. The general trend of increasing incidence with annulus height is due to the use of a blade inlet angle which is constant with radial height. This trend could be eliminated if free-vortex blading was used. The localised increase in incidence near to the walls is due to the low axial velocities in those regions. The greater thickness of the outer wall boundary layer (as compared to the inner wall layer) is revealed by the greater radial depth of this near-wall increase. The Mellor-NACA cascade data (Wilson[2]) suggests that for similar blading to this 'low stagger' geometry, the stall limit is at approximately 20° incidence. Therefore it seems unlikely that the rotor should be stalled in these near-wall regions.

4.1.2 Flow Conditions Between Rotor and Stator

A single radial traverse was performed at this station using a five-hole probe, with the probe tip inclined into the flow at an angle equal to the stator inlet angle. For the 'low stagger' blading, the blade inlet angle (30°) is the same for rotor and stator since the degree of reaction is 50% (see Appendix 1 for blade design details). This angling of the probe was done in order
to maintain accuracy by ensuring that the test data lay close to the centre of the probes calibration range. When mounted in the test rig in this fashion, the axial location of the probe tip was approximately mid-way between rotor trailing edge and stator leading edge (Fig.2.3). At this station and at all subsequent stations downstream, measurements were taken for both the normal straight stator blade, and for the chevron stator blade.

The axial velocity profile of Fig.4.1.3 reveals that no major changes in flow distribution have taken place in the passage of the air through the rotor (c.f. Fig.4.1.1). There is however a noticeable thickening of the boundary layer near the inner wall due to the influence of several factors:

i) rapid growth of a relatively thin boundary layer. This is to be expected from the degree of shear present in this layer at Station 2.

ii) rotor incidence effects. It is possible that the higher incidence near the wall causes the localised suction surface boundary layer to be thicker.

iii) secondary flow effects. These flows tend to redistribute fluid and in so doing increase the boundary layer thickness by moving low energy fluid away from the walls, whilst moving higher energy fluid nearer to the wall where it subsequently loses energy due to high shear forces.

Near to the outer wall the situation is somewhat different, with rotor tip leakage flows having a significant effect, and the boundary layer thickness staying approximately the same. It is interesting to note how the gradient of the velocity profile decreases in the region 90% to 100% blade height, indicating an increase in the boundary layer displacement thickness and shape parameter as confirmed by Table 4.1. This appears to be an effect arising from the influence of the tip leakage flows, since it is not observed near the inner wall.

The calculated distribution of rotor deviation angle is shown in Fig.4.1.4. Near the inner wall the classical overturning of the fluid attributed to blade passage secondary flows is apparent. This type of fluid movement has been documented by Horlock[3] and many others. The overturning immediately adjacent to the wall is accompanied by underturning (relative to the flow at mid-span) at a slightly greater immersion from the inner wall.

In the region near to the outer wall, there appears to be a similar pattern emerging from about 75% to 90% blade height, but then nearer to the wall this effect is swamped by severely underturned fluid associated with tip leakage flows.

The difference in level of these two curves across the main part of the span of about 1.5° is considered to be due to a slight difference in setting the probe inclination angle for the two runs. The accuracy of setting the probe angle is to within 2°. This same difference in level between the two curves is evident once again in Fig.4.1.5 which shows the radial variation of OGV incidence.

The deviation at mid-blade height is approximately 2° (Fig.4.1.4), which compares with the estimated value based on cascade data of 6°. The calculation of the rotor deviation uses
measured values of flow velocity and angle, and also the blade speed. Checks were carried out to assess the accuracy of these quantities and showed that measurement errors could not account for this discrepancy. Ravindranath & Lakshminarayana[56] have reported a similar discrepancy between observed rotor deviation and cascade correlation, of about 4 - 5°. They attribute this difference to the inability of cascade correlations to deal with 'three dimensional viscous and inviscid effects'.

By consideration of the blade velocity triangles it can be shown that the OGV incidence is dependent on both rotor deviation and axial velocity. Negative deviation angles and low axial velocities have an additive effect of increasing the OGV incidence near the inner wall. In order to predict the stalling incidence cascade data can be used although because of three-dimensional effects and higher turbulence levels present in rotating turbomachinery these predictions should only be treated as approximate. Since the stage reaction is 50% and the space/chord ratio is similar for both rotor and stator then the same stall limit criterion applies, i.e., about 20° of positive incidence. Fig.4.1.5 shows that the only region where the stator incidence reaches this level is very close to the inner wall. The variation of incidence near the outer wall is of interest in the way in which it increases from 80% to 90% blade height, and then decreases closer to the wall. This decrease is attributed to the influence of tip leakage flows, as already observed from the axial velocity profile and the variation of rotor deviation.

4.1.3 Flow conditions at OGV Exit

The results from the area traverse performed downstream of the OGV's are presented in several different forms. Fig.4.1.6 shows the pitch-averaged velocity distribution at this station for both the straight and the chevron blades.

The velocity profiles reveal further thickening of both inner and outer wall boundary layers, although the shape parameter is reduced (Table 4.1), and also a noticeable mass flow deficit at mid-height for the chevron blades. Thickening of the wall boundary layers is a well documented phenomenon which has been observed by many researchers in the past e.g Smith[88], Howell[87]. The behaviour of the wall boundary layers in axial compressors is affected by a large number of parameters; e.g. diffusing nature of blade passages, distance along the wall traversed by the flow, blade boundary layers and wakes and their interaction with the wall shear layers, turbulence levels, blade tip leakage flows, sheared and skewed inlet flow; and hence it is difficult to isolate from a simple axial velocity plot the degree to which any one of these parameters has affected the inlet flow.

The noticeable reduction of mass flow at mid height for the chevron blade has a radial position coincident with the centre bend of the chevron shape. Therefore it seems likely that this mass flow deficit is associated with a suction surface corner effect, where the boundary
Results and Discussion

layers from the two halves of the blade suction surface interact.

A rather more detailed assessment of the variation of the axial velocity at this downstream station can be achieved by examination of the contours of axial velocity as shown in Fig.4.1.8. The most obvious feature which differentiates the two plots of Fig.4.1.8 is the OGV blade shape, with the wakes showing very clearly the straight and chevron geometries. In the case of the straight blade the wakes can be seen to be well defined but quite narrow, thus suggesting no flow separation from the suction surface. It is of interest to note however that the flow exiting from the suction surface-outer wall corner seems to be retarded rather more than in the corresponding corner near the inner wall. It seems unlikely that a stall would occur since it has already been noted (Section 4.1.2) that the OGV incidence in this outer wall region is about 10° or so less than the estimated stalling incidence. However, it is obvious that significant boundary layer growth has taken place in this corner.

A similar observation can also be made from the axial velocity contours for the chevron blades (Fig.4.1.8a). Also visible from this plot is the more pronounced wake at blade mid height, which is the cause of the mid height mass flow deficit of Fig.4.1.6. Mojola & Young[118] note that corner flows subjected to an adverse pressure gradient will separate quite readily especially if the boundary layers are laminar or transitional. However the mid-height axial velocity contours of Fig.4.1.8a do not suggest any flow separation since the increase in wake width and depth at this radial location is relatively minor. Mojola & Young also report that the direction of the corner flow reverses as the boundary layer changes from laminar to turbulent (Fig.1.7). Therefore the direction of rotation of the fluid at this point should provide some indication of the state of the corner boundary layer. However the plots of velocity vectors at each data point (Fig.4.1.9a) do not reveal any hint of fluid rotation in this region. It seems as though the corner effect seen on the chevron blade is not so much caused by the action of secondary flows but rather by a simple coalescing of the two boundary layers. The reasons why secondary flows are not apparent are; the large corner angle of 120° (c.f. 90° used by Mojola & Young [118], Zamir & Young[24]), the short corner length, and also because the 'corner' on the chevron blade suction surface is radiussed rather than being sharply defined.

From Fig.4.1.6 it is clear that there are differences in the mass flow distribution between the ordinary straight and the chevron blades, but significantly no major improvement in the near inner wall flow as had been reported by Harasgama[81] (Fig.4.1.7). On closer comparison with Harasgama's results it can be seen that his velocity profile for straight blades was in poor shape near the inner wall, whereas for the tests reported here no such feature is apparent. Harasgama's axial velocity contours revealed that the cause of the poor inner wall flow was a region of separated fluid in the suction surface - inner wall corner. When he tested his chevron blades, this region of separated fluid was found to have been eliminated thus producing an improved velocity distribution. In the current tests, since no separation region
Results and Discussion

was observed for the straight blades, there is no possibility of a similar type of improvement to
the flow. The reason why the separation region is not seen in the current tests is discussed in
Section 4.3.2.

Since the mean velocity vector is fully defined at each data point, then the velocity
components in the plane normal to the axial direction can be analysed, in addition to the axial
velocity discussions above. This data is presented in a qualitative manner in the flow vector
plots of Fig.4.1.9. In both plots the length of the arrows is non-dimensionalised by the mean
axial velocity and then scaled to give a suitable average length.

Perhaps the first main feature to be noted from these plots is the positions of the blade
wakes as indicated by abrupt changes of velocity vector. These regions correspond to the
wake regions depicted in the axial velocity contours of Fig.4.1.8. It should be remembered at
this point that this traverse station is a short way \(x/C_{ax} = 0.128\) downstream of the OGV’s,
and therefore whilst there is still significant mixing occurring within the wakes, the most
vigorous mixing in the wakes (at the trailing edge region) will not be present in these plots.

A further consequence of the downstream distance of the traverse station is that the probe
is within the diffuser. This is evidenced by the large radial velocities in an outward direction
which are especially pronounced in the region near the outer wall. This is simply due to the
fluid tending to follow the wall geometry. This radial velocity will clearly vary according to
radial positioning, and should be borne in mind whilst further analysis of the secondary flow
structure is performed. Fig.4.1.10 shows the same data as Fig.4.1.9, but with the radial
velocity component associated with diffuser curvature removed. Also to be borne in mind is
the possibility that the secondary flows within the stator blade passages might well have
experienced significant decay by the time the flow reaches this traverse station. This point
highlights the difficulty of attempting to determine what is happening within the blade passage
by studying the exit flows, and emphasises the need for flow measurements to be made within
the flow passage in order to properly understand the inter-blade flows.

For both the chevron and straight blades there is evidence of secondary flow in the regions
close to the end-walls. Looking firstly at the straight blades (Fig.4.1.9b), near to the inner
wall, there is a movement of flow toward the pressure side corner which is then swept across
the end-wall toward and then part way up the suction surface where it becomes entrained in
the suction surface boundary layer. A similar effect is seen in the region close to the outer
wall, although the flow is modified to some extent by the diverging outer wall and also the
distorted casing as mentioned above.

The corresponding flow vector plot for the chevron blades (Fig.4.1.9a) reveals basically
the same near-wall secondary flow patterns, though in both the suction surface-endwall
corners the secondary flows seem to be somewhat less pronounced. This trend is perhaps
better observed in Fig.4.1.10. A similar reduction in secondary flow strength has been
observed by Shi Jing et. al. [75]. It is apparent that there is a general increase in residual flow
swirl when compared with the straight blades, this being especially noticeable in the outer half of the annulus. This difference in magnitude of circumferential velocity is shown in a simpler form in Fig.4.1.11. This plot of pitch-averaged yaw angle also shows the net effect of the classical blade passage secondary flows i.e. the flow overturning close to the end-walls. This increase of flow turning is clearly evident near the outer wall where substantial flow angle changes take place; for the outer 15% or so of the annulus the flow angle change is of the order of 7-10°. There is noticeable underturning of the fluid, in both cases, in the region around 80% annulus height. This is a feature often observed when secondary flow overturning occurs closer to the wall (Horlock[3]).

Near the inner wall, a less clear pattern emerges. For the straight blade case the flow is overturning at about the 3% annulus height position, but closer to the inner wall the amount of overturning decreases. This is contrary to the classical secondary flow observations (e.g. Horlock[3]) which shows that the amount of overturning increases rapidly close to the wall. This apparent anomaly arises due to a reversal of the secondary flow directions in the region very close to the inner wall and toward the suction surface (Fig.4.1.9b). The point at which this direction change takes place experiences a localised accumulation of low energy fluid (Fig.4.1.9b). The reason why this secondary flow reversal should take place is not clear, though it may well be connected with the very high OGV incidence in this region very close to the inner wall (Fig.4.1.5).

For the chevron blade, near the inner wall, Fig.4.1.11 shows very little mean flow angle change in either the under or over-turning directions. From the vector plot (Fig.4.1.9a), the flow in the region very close to the inner wall experiences a similar direction change, though perhaps less pronounced, which produces a very small net mean secondary flow. Slightly further out from the inner wall, at about 3-5% annulus height, there is no pronounced overturning as seen for the straight blades (Fig.4.1.9b).

The large difference in level of pitch-averaged yaw angle between the two blade types of mean circumferential velocity is the result of reduced flow turning with the chevron blades. The increased level of swirl with the chevron blades can also be concluded from the way in which the blade wakes move circumferentially down the diffuser as shown by Fig.4.4.7 (c.f. the corresponding plots for the straight blades in Fig.4.4.8). Whilst Harasgama[81] does not present a comparison of OGV exit yaw angles between his chevron and straight OGV’s, nevertheless the fact that similar high levels of residual swirl were encountered with his chevron blades can be extracted from the circumferential wake movement as revealed by his diffuser axial velocity contour plots.

Table 4.3 summarises the performance of the two blade types in quantitative terms. The kinetic energy flux coefficient ($\alpha$) at OGV exit is significantly higher for the straight blade, implying a rather more peaked velocity profile. The calculation of $\alpha$ includes the effect of velocity variations in the circumferential direction as well as the radial direction. At the station
Results and Discussion
downstream of the OGV's the main factor causing circumferential variations of axial velocity is the presence of OGV wakes. However from Fig.4.1.8 it can be seen that the wakes are of similar proportions for both blade types, with the exception of the mid-height region where the chevron blade wake is more severe. This would cause $\alpha$ for the chevron blade to be higher rather than lower than that for the straight blades. Therefore the reason for the difference in $\alpha$ must lie with the radial variation of axial velocity, which is indicated most clearly by Fig.4.1.6. Whilst the mid-height velocity deficit for the chevron blade acts to increase $\alpha$, the reason for $\alpha$ being higher for the straight blade can be quite clearly attributed to the very low velocities near the outer wall, which have been noted before and are due to a casing distortion problem. For this reason the apparent improvement in $\alpha$ for the chevron blades should be treated with some caution.

Similar points can also be made concerning the OGV loss figures, where it is quite possible that the OGV loss for the straight blade is somewhat higher than it would otherwise have been if the casing distortion problem had not been present. The level of loss for both blade types (11% - chevron, 12% - straight) is somewhat higher than the predicted loss of 5% (Appendix 1). This difference is due to the under-estimation of the losses associated with the inlet boundary layers and the attendant high incidences, and also because the figures include some loss due to mixing out of rotor and stator blade wakes, since the traverse stations do not coincide with the axial locations of the blade leading and trailing edges. It is thought highly unlikely that the relatively high levels of measured loss (relative to predicted levels) are due to a Reynolds number effect since the blade chord Reynolds number is quite high ($1.57 \times 10^5$), and also the blade boundary layers are subject to high levels of unsteadiness due to rotor wakes which serve to decrease the critical Reynolds number. Harasgama tests [811], conducted on a similar blade but at a lower Reynolds number ($0.4 \times 10^5$), indicate a loss of approximately 18%. This comparison implies that the Reynolds number for the tests reported here is above the transitional level. Reynolds number effects are further discussed in Section 4.3.2.

The regions of the stator exit flow which contain the high loss fluid can be revealed by plots of contours of constant loss, as given in Fig.4.1.12. These two plots clearly show the high loss associated with the wakes from the OGV's, and how this wake loss is greater at mid-height for the chevron blades. Near the inner wall, in the inter-blade region, there is slightly increased loss associated with wall boundary layer development and secondary flow effects. It is interesting to note that for the straight blade this seems to be more associated with the suction surface corner, whereas for the chevron blade it appears to be more concentrated at the pressure surface corner. This feature suggests a difference in blade passage secondary flow, and Fig.4.1.10 shows that there is a stronger flow along the inner wall toward the suction surface for the straight blade. Near the outer wall a considerably different picture emerges. In both cases in the pressure surface corner a region of negative loss is apparent. It is thought that this region has been caused by the combination of two phenomena; blade
passage secondary flows, and rotor tip leakage flows. Rotor tip leakage fluid is a highly
turbulent type of flow and since vigorous mixing occurs within it therefore the entrainment of
higher energy fluid from the main part of the blade passage is a strong possibility. (Turning to
the secondary flow aspect, Fig.4.1.9 shows considerable flow movement from within the
blade passage toward the outer wall pressure surface corner, thus bringing high energy fluid
into that corner. In the region of the outer wall suction surface corner it appears that there is
substantial evidence of secondary flows, with low energy fluid being swept part way along
the suction surface of the blade. This low energy fluid is likely to be that which was
previously associated with the outer wall boundary layer/
4.2 'High Stagger' Compressor Test Results

4.2.1 Rotor Inlet Conditions

The test rig configuration for these test runs differs from that for the 'low stagger' tests as detailed in Section 2.4 but the main differences are as follows:

i) Air filters are installed in the inlet plenum chamber. See Appendix 2 for specification and arrangement.

ii) The full length of annular inlet to the compressor is used (Fig.2.1).

iii) Different compressor blading. See Appendix 1 for blade design details.

The use of the full-length annular inlet causes quite a significant change in the nature of the inlet velocity profile. Fig.4.2.1a shows the velocity profile at this station. In comparison to that for the 'low stagger' tests (Fig.4.1.1), the velocity profile is much more peaked with the consequent very thick wall boundary layers. This well sheared profile is a result of the long inlet length over which the boundary layers develop. It will be observed that the profile has a noticeable hub-bias, which is a consequence of the outwardly diverging diffusing section located a short distance upstream of this station; a phenomenon reported by others (e.g. Stevens & Williams[102]).

The velocity profile achieved in the latter stages of two different multi-stage compressors is shown in Fig.4.2.1b from the data of Smith [88] and Howell [87] and, whilst these two profiles differ in shape, they both show a high degree of shear. The velocity profile due to Howell [87] shows very good agreement with the inlet velocity profile for the 'high stagger' blades.

Fig.4.2.1a compares the inlet axial velocity profile for the two test runs; for chevron and straight OGV's respectively. As expected, the change of stator blade type has negligible effect on the velocity profile at this upstream station. The cause of the data scatter seen on these two curves, particularly in the region 20% - 80% annulus height, is not clear. The boundary layers are 'tripped' at an early stage in the entry length and can be considered fully turbulent, thus ruling out any unsteady transitional effects as being the cause.

Fig.4.2.2 shows the radial variation of the flow incidence onto the rotor blades, calculated from the axial velocity distribution, the rotor speed, and the blade inlet angles. It can be seen that the mean level of rotor incidence is approximately 1°, which is close enough to the design incidence of 0° that nominal design conditions for the compressor can be said to have been achieved. The negative mid-height incidence and positive near-wall incidences closely reflect the radial variation of inlet axial velocity as observed already in Fig.4.2.1. The actual level of incidence close to the walls is of the order of 7 - 8°. An estimate of the stalling incidence for this blading has been made from the cascade data (Horlock[3]), of +6°. On this basis it might be expected that flow separation would occur from the suction surface at these regions close to the wall. However, since these high incidence conditions are very localised, it is possible that
flow separation will be inhibited by three-dimensional effects.

4.2.2 Flow Conditions Between Rotor and Stator

The axial spacing (14mm) between rotor trailing edge and stator leading edge is quite small for the 'high stagger' test build. This means that even though the traverse boss was situated as close as possible to the OGV leading edge, the tip of the five hole probe was nevertheless very close to the rotor trailing edge (Fig.2.4); this distance, non-dimensionalised by the rotor blade axial chord is 0.074. This point should be borne in mind when analysing the traverse results.

Fig.4.2.3 gives the axial velocity profile at this station for the two OGV types. The only point of note at which the two curves differ is in the outer 5% of the annulus. However this difference is only very minor and is thought to be caused by a small difference in rotor tip clearance between the two runs brought about by a difference in the temperature of the inlet air.

It is obvious on comparison of Fig.4.2.3 with Fig.4.2.1 that a substantial change in mass flow distribution has taken place within the rotor blade passages. The velocity profile has become less peaked, although a considerable reduction in mass flow has been incurred near the outer wall. It is likely that this region of low velocity is attributable to tip leakage effects, although it is not clear whether the large reduction of mass flow is due merely to increased mixing of the tip leakage flow with the passage flow, or due to a rotor tip stall (perhaps triggered by tip leakage effects) combined with tip leakage flows. A localised stall at rotor tip would be possible since it has already been noted in Section 4.2.1 that the incidence level in this region is slightly higher than the estimated stalling incidence.

Whilst the axial component of the velocity is very low near the walls, the magnitude of the absolute velocity (Fig.4.2.4) does not show such a marked change. The general level of the absolute velocity is obviously higher than the axial velocity because the flow is swirling. Near the outer wall there is a marked drop in velocity, though not as severe as for the axial velocity. Near the inner wall the absolute velocity shows little change from the mean level across the annulus. These relatively high absolute velocities near the walls coupled with the low axial component, imply high swirl angles in these near wall regions.

Fig.4.2.5 shows the radial distribution of rotor deviation angle for both stator blade types. It is immediately apparent from this figure that there is a difference in level between the two curves of about 2°. It is thought that this difference is a result of inaccuracy in measurement of the velocity for the chevron OGV case (for this station the mass flow error for the chevron case being +4%), since the velocity is used in order to calculate the deviation. Hand calculation shows that if the mass flow error for the chevron OGV test was lower, then the respective curve of Fig.4.2.5 would correspond more closely to that for the straight blade test
Results and Discussion

(for which the mass flow error is less than 1%). Therefore, the following comments relate to the curve for the straight blade test only, since that curve is more representative of the true situation.

The level of rotor deviation over the range 20-50% of annulus height is 2-3° lower than that suggested by the cascade-based prediction of about 5° (see Appendix 1). Ravindranath & Lakshminarayana have reported similar observations downstream of a compressor rotor, quoting measured deviation angles 40-50% lower than cascade-based predictions. They attribute this difference to the inability of cascade correlations to deal with 'three-dimensional viscous and inviscid effects', but do not expand the point.

It is interesting to note the difference in the deviation near the inner wall between this 'high stagger' test and for the 'low stagger' test. In Fig.4.1.4 negative deviation angles can be seen near the inner wall which can be explained by classical secondary flow arguments. However, for the 'high stagger' test the opposite trend is seen i.e. increased positive deviation near the inner wall. It has already been pointed out in Section 4.2.1 that the rotor incidence is slightly higher than the estimated stalling incidence in very localised regions close to the end-walls. Therefore it seems likely that the reason for the high deviation angles close to the inner wall (Fig.4.2.5) is a small hub stall. Such a stall would disturb the secondary flow structure, and from Fig.4.2.5 it appears that any classical type secondary flow near the inner wall has been completely nullified by this hub stall. For the 'low stagger' tests (Fig.4.1.4) a normal secondary flow pattern is observed as the near-wall rotor incidence for these tests was well within the estimated stalling incidence and therefore no hub stall occurred. The reason why the near-wall incidences for the 'high stagger' blades are so much higher is due to the higher degree of shear in the inlet velocity profile and that whilst the 'high stagger' blades have an optimum spacing (i.e. the minimum number of blades to achieve the necessary flow turning without separation), the 'low stagger' blades are about twice as closely packed as the optimum.

Fig.4.2.6 shows the radial variation of the flow incidence onto the OGV's. The existence of highly swirling fluid near to both walls is confirmed in this figure by the very high levels of positive incidence in those regions. The data for the two cases of stator blade type agree very well, with no difference worthy of any note. It will be noted that in the region 20% - 70% of blade span that the OGV incidence is negative, and when this is weighed against the positive near-wall incidences it will be seen that a reasonable compromise has been reached in attempting to achieve nominal design conditions (zero incidence).

For the straight blade case only, a further traverse was performed, with a cylindrical three-hole yawmeter probe. Using this instrument, measurement of the flow incidence onto the OGV's could be made at a location closer to the OGV leading edge than was possible with the five-hole probe. The resulting axial velocity distribution is seen in Fig.4.2.3, from which it is apparent that a significant increase in near-outer wall velocity has occurred between the
Results and Discussion

two traverses. This local increase in mass flow is balanced by a decrease in the region from 20% - 80% annulus height. The suggested cause of this change of flow distribution is intense mixing associated with the decay of rotor wake and tip leakage flows in the outer wall region. Lakshminarayana & Ravindranath[55] have showed that the tip leakage fluid plays a major role in this region.

In Fig.4.2.6 the incidence is plotted for the three-hole probe traverse alongside the data from the five-hole probe. It can be clearly seen that whilst there is good agreement in that part of the span from 10% - 70% height, outside of those regions some decay of the amount of flow swirl has occurred. Again, this is thought to be a feature of the mixing out of large wake/tip leakage flows. Ravindranath & Lakshminarayana[56] have also observed decay of swirl angles immediately downstream of their rotor, which they attribute to rotor wake effects. As a result of their observation Ravindranath & Lakshminarayana suggest that an axial spacing of at least one blade chord should be used between rotor and stator if unacceptably high stator incidence is to be avoided. Since the level of swirl decay can be calculated (using the traverse data from the three-hole and five-hole probes), then an extrapolation can be performed to the stator leading edge to provide an estimate of the true OGV incidence. This estimate is indicated by the dotted line on Fig.4.2.6, and indicates very significantly reduced stator incidence from that suggested by the data from the five-hole probe in both the wall regions, but especially so in the outer wall region. Cascade predictions suggest a stalling incidence of about +14° for these stator blades, and therefore it can be seen that in the near wall regions the blade will be operating just within the operating limits.

This feature of swirl decay is of great importance in rig tests involving measurement of rotor/stator performance since the results will be dependent on the axial positioning of the probe relative to the blade row. In particular when the upstream flow is exiting from a rotor blade row then any measurements of a downstream stator row should be done with the inlet traverse performed as close as possible to the leading edge of the stator row.

The engine designer will also have considerable interest in this swirl decay, since it implies that if he chooses to minimise engine length by reducing spacing between blade rows then performance will suffer due to localised regions of very high incidence. However it is interesting to note in this connection that Smith [88] indicates that reduced axial spacing leads to improved pressure rise over the workable range of flow coefficient, reduced deviation angles and reduced losses, which he speculates to be due to time-unsteady effects which arise because the downstream blade row exerts some degree of upstream influence on the blade row moving relative to it. Koch & Smith [89] indicate that the end-wall boundary layer becomes thinner for smaller axial spacings, and Koch [119] indicates that a reduced axial gap leads to improved stalling pressure rise. The point at which reduced axial spacing begins to have a beneficial effect is given by Howell [87] as about 1/6 of a blade chord, and by Koch [119] as about 1/2 of the tangential blade spacing.
4.2.3 OGV Flow Conditions

Tests performed in the region between leading and trailing edge of the OGV consisted of blade surface pressure tappings, and (for the straight blades only) inter-blade traverses with a custom-built five-hole probe, and wool tuft flow visualisation. The bulk of the information presented in this sub-section is connected with the inter-blade traverse results and therefore discussion is centred upon the straight blades.

In order to get some sort of general overview of the way in which the blade passage flow progresses, the radial distribution of mass flow is given in Fig.4.2.7 in the form of the pitch-averaged velocity profiles. Fig.4.2.7a shows that a reasonably uniform rate of change in flow distribution occurs throughout the passage, but with slightly higher rates of change occurring in the first and last 10% regions of the passage. The slightly higher rate of change within the first 10% of the blade passage which is apparent near the outer wall is most probably due to continuing rotor wake mixing effects. Fig.4.2.7b gives a comparison of the velocity profile at three chordwise positions with those at the W4 and W5 stations (i.e. the stations immediately upstream and downstream of the stator row). This figure shows perhaps a little more clearly that a general fluid movement occurs from the lower radii to the region close to the outer wall. This is in some part due to mixing out of the large rotor wake/tip leakage flow region near the outer wall. Another factor which would tend to cause this trend is the radial pressure gradient due to the presence of the diffuser which acts at OGV exit in such a manner as to encourage fluid to move to higher radii. It can be seen that whilst the agreement between the 100% chord profile and the W5 (i.e. downstream traverse) profile is fairly good, there is a marked difference between the W4 (i.e. upstream traverse) profile and the 0% chord profile. This large difference at inlet to the stator blade row is due to the intense mixing immediately downstream of the rotor row as discussed in Section 4.2.2.

Fig.4.2.8 presents relief maps of the passage velocity at every 10% of stator blade chord. These plots, together with those of Fig's 4.2.9, 4.2.10, 4.2.11 are produced using interpolated data, and therefore the nodes of the mesh on each of the maps are interpolated points, i.e. they are not the original data points. The original data point locations can be ascertained from the plot of flow vectors (Fig.4.2.12). Several features can be picked out from Fig.4.2.8:

i) There is a progressive reduction in mean velocity level through the blade passage, as would be expected for a diffusing passage. Over the first 30% or so of the blade passage, the mean velocity level remains fairly constant since the chordwise pressure gradient is low (Fig.4.2.11) because the diffusion caused by the change in swirl is counteracted by the constriction of the flow passage as the blade thickness increases from zero at the leading edge to maximum thickness at 30% chord.

ii) For the majority of the chordwise planes it will be observed that there is a velocity
Results and Discussion

gradient between blade surfaces with the higher velocities being closer to the suction surface. This higher suction surface velocity is due to the flow acceleration experienced over the convex side of the aerofoil surface. At the 100% chord position, this velocity gradient has decreased to a negligible level.

iii) A number of minor features are also present. The somewhat high rate of change of pitch-averaged velocity profile over the last 10% chord near the inner wall which was noted from Fig. 4.2.7a, can be traced to a localised thickening of the suction surface boundary layer at that radial location. Midway between the blades, near the inner wall, a region of slightly lower velocity fluid develops which is still present at the 100% chord position. This is a result of secondary flow movement within the blade passage, in the form of low energy fluid from the blade suction surface being swept along the inner wall toward the pressure surface (Fig. 4.2.12). The reason why this low energy fluid region does not progress all the way toward the pressure surface is that the flow just described reduces in strength and a flow in the opposite direction is set up later in the passage (Fig. 4.2.12).

Fig. 4.2.10 shows a further selection of relief maps, this time tracing the behaviour of the flow stagnation pressure. The data near the walls in these plots is based on a linear extrapolation of the data further away from the walls. An approximate assessment of the degree of extrapolation required for each plot can be made from Fig. 4.2.12 which shows the positions of all the original data points and the relative wall positions.

The distribution at 0% chord is largely flat, due essentially to a fairly flat velocity distribution (Fig. 4.2.8). Near the outer wall however, there is a noticeable dip in the total pressure, which is due to the mixing out of the rotor wake and tip leakage flows in that region. This region of reducing total pressure shows continued decrease within the first 10% of blade chord, and continues to be present throughout the blade passage, although its prominence is decreased in the latter chordwise locations presumably due to redistribution of the low energy fluid within the blade passage.

Near the inner wall a deficit midway between the blades develops and seems to remain in that position. This feature correlates well with a low velocity region noted from Fig. 4.2.8. Whilst it is clearly correct to anticipate a correlation between the plots of velocity and stagnation pressure, some apparent discrepancy may arise since:

i) The velocity maps indicate the velocity component in the direction parallel with a tangent to the blade camber at that traverse plane rather than the absolute velocity.

ii) Losses in the boundary layers and also as mixing occurs.

In both the suction surface-end wall corners, very localised regions of low energy fluid begin to develop, which grow progressively in the chordwise direction and correlate well with the velocity distributions of Fig. 4.2.8.
The static pressure variation is given in Fig.4.2.11. The main features to be noted from this figure are as follows:

i) Because the blading is of a diffusing nature the mean level of static pressure rises through the flow passage. This increase of pressure corresponds well with the decrease in velocity of Fig.4.2.8, where most of the change occurs in the region 40% - 100% chord.

ii) Over the first 30% of the blade chord a positive radial pressure gradient is discernible, i.e. the pressure increases with radius in this region. This gradient is set up due to the swirl present in the flow at exit from the rotor as the fluid tends toward radial equilibrium, and therefore as the swirl is progressively reduced through the stator blade passage, so the radial pressure gradient reduces. At exit from the blade, some evidence of a negative pressure gradient was expected due to the radially outward flow path at inlet to the diffuser. However no such evidence is apparent from Fig.4.2.11, the reason being that the gradient is too shallow to be readily visible on such a plot.

iii) A pitchwise pressure gradient is present over the major part of the blade chord, and is particularly noticeable at the 0% chord plane. It is this gradient which causes the fluid to follow the camber of the blades and thus reduce the flow swirl.

Since this pressure gradient is at a high level at the 0% chord position, it will continue to be measurable for a short distance upstream of the stator leading edge. Therefore the traverse performed upstream of the OGV's should take the form of an area traverse, rather than a single radial traverse, in order to accurately assess mean inlet conditions.

At the 0% chord plane the pressure adjacent to the pressure surface is higher in the wall regions. These wall regions are those where the flow swirl is particularly high (Fig. 4.2.6), and therefore a steeper pressure gradient is required to 'persuade' the fluid to follow the blade passage camber.

Fig.4.2.12 shows the flow vector at each data point within the blade passage. Whilst the planes on which these data points lie are horizontal (see Fig.2.6) the flow vectors are not resolved into these planes, but rather each flow vector is resolved into the plane perpendicular to the camber line at that point. In effect what this means is that if the flow were to follow the camber line throughout the blade passage then the vector plots would only show a dot at each data point. Therefore any flow movement shown on the vector plots indicates the presence of some form of secondary flow or flow separation etc.

The vector plot at 0% chord shows considerable secondary flow movement, thus providing justification for the suggestion made above in the discussion of Fig.4.2.11 that the presence of the blade passage may have considerable upstream influence on the flow at the upstream traverse station. There is a general trend for the flow to move away from both blade surfaces at this chordwise location. This movement is due a 'funnelling' of the flow into the blade passage, or put another way, it is the result of flow accelerations around the leading
edge of the blades toward the centre of the blade passage. This effect is also present, though to a lesser extent, at the 10% and 20% chord positions.

A further feature superimposed on the secondary flow structure at 0% chord, is the flow movement due to the radially varying incidence. This feature can be clearly seen in the region mid-way between blade surfaces. In the regions 0-5% and 75-100% height where the incidence onto the blades is positive (Fig.4.2.6) the flow still has momentum in the direction of the swirl i.e. toward the stator blade pressure surface. This contrasts with the region from 5-75% height where the incidence is (slightly) negative and the resultant flow movement is in the opposite direction i.e. toward the suction surface. The net result of this is the formation of two vortices which rotate in the opposite sense to classical curved duct secondary flows. The reason for this is of course due to the radially varying incidence. In the vast majority of studies which have investigated blade passage secondary flows, there has been no simulation of this incidence variation. Boletis et. al.[30] and Carrick [29] have reported experiments in turbine blading operating with inlet skew, but since the inlet skew operates in the opposite sense for turbine blading therefore their results cannot be compared with the results reported here, and also their tests did not include any traversing within the blade passage. No previous work for compressors operating with inlet skew in which blade passage traversing was performed has been found, and therefore this type of vortex structure has not been reported before.

This vortex structure continues to be present further along the blade passage, although its strength diminishes such that at the 50% chord position there is no discernible net movement of flow from pressure to suction surface at the mid-height location. The regions of underturning fluid continue to be present however, and appear to move away from the walls slightly. This movement away from the end-walls is perhaps due to the formation of 'classical' secondary flow vortices very close to the walls in the very low energy fluid associated with the annulus wall boundary layers. These wall vortices serve to strengthen the underturning of the inlet vortices as shown in Fig.4.2.13. It is difficult to ascertain the strength of these wall vortices because of the lack of data points close to the walls. Also, the large flow arrows in the latter stages of the blade passage, indicating the normal deviation of the fluid in a blade row, tend to mask the wall vortices. Nevertheless, it can be seen at the 100% chord position that the amount of turning is slightly reduced at those data points closest to the end-walls, implying that overturning is present closer to the end-walls. A broader view of the development of the blade passage secondary flows can be attained from Fig.4.2.14 which shows the pitch-averaged yaw angle variation throughout the blade passage. Near the OGV leading edge Fig.4.2.14 indicates the mid-passage overturning and near-wall underturning associated with the radial variation of flow incidence. As the flow progresses through the blade passage the regions of localised underturning tend to move away from the walls as the classical type of secondary flow develops with overturning (or reduced underturning) close in to the walls. A further obvious trend apparent from Fig.4.2.14 is the
tendency for the mean level of underturning to increase through the blade passage. The main reason for this is the rapid boundary layer growth on the blade suction surface in comparison to that on the pressure surface. However, the mean level of underturning at 100% chord (about 13°) is greater than the estimated level of deviation (8°). Further comment on this point is given in Section 4.2.4.

Fig.4.2.15 shows the blade surface static pressure distribution for both chevron and straight stator blades, the data having been obtained by measurements from surface pressure tappings. These tappings were located at 10%, 50%, and 90% annulus height for several chordwise locations. Because of difficulties with the routing of the necessary pressure tappings and tubing, the tappings were distributed over six blades for each blade type as detailed in Table 2.1. Looking firstly at the distribution for the straight blades (Fig.4.2.15b), there are several features which can be noted. Before discussion of these features it should be pointed out that there are a number of factors which influence the surface pressure distribution and since these factors all operate simultaneously it is sometimes difficult to isolate the effects of individual factors. However, observations which can be made are as follows:

i) The effect of radial variation of incidence. Masek & Norbury[6] have reported pressure distributions for a cascade (essentially two-dimensional) under varying incidence conditions. They have showed that upon severe negative incidence the flow eventually separates from the leading edge on the pressure surface, whereas for positive incidence conditions the suction peak on the suction surface moves forward leading eventually to suction surface separation. At modest positive incidence the separated fluid from the suction surface was observed to re-attach, the length of the separation bubble being determined by the severity of the incidence. In Fig.4.2.15b the effect of the incidence variation seems to be confined to the region close to the leading edge i.e. up to about 20% chord. At 10% annulus height the incidence is only slightly positive and therefore the pressures at this location provide a basis for comparison. At 50% height a negative incidence is encountered, which is reflected by the rearward shift of the suction peak on the suction surface. The value on the pressure surface seems to be little changed from that at 10% height due to inhibiting of any pressure surface separation by the rounded leading edge. At 90% height the incidence is positive, which is confirmed by the forward shift of the suction peak and a high initial pressure on the pressure side of the blade. Masek & Norbury showed that separation from the suction surface is characterised by an abrupt 'flattening' of the suction surface distribution, with the new level of pressure being maintained to the trailing edge. They also showed that the pressure side distribution is affected with a fall in pressure to the same level as on the suction surface at the trailing edge. This form of distortion of the pressure distribution is not seen in Fig.4.2.15b at any of the three radial positions, indicating that there is no suction surface separation. This was confirmed from flow visualisation tests with wool tufts mounted on the blade surfaces at
similar locations to the pressure tappings.

ii) Radial variation of static pressure both at inlet to and at exit from the stator blade row is present. At inlet a positive gradient arises due to a tendency of the flow toward radial equilibrium with high swirl. At exit from the blade row a negative pressure gradient exists due to the presence of an outwardly canted diffuser. These two gradients are therefore acting in an opposite sense, and combine to produce a more severe axial pressure gradient at the hub than at the casing.

Fig.4.2.15b shows that in the latter part of the blade, there is little evidence of any radial pressure gradient, indicating that the gradient due to diffuser curvature is only small. In the first 70% of the blade chord however, there is evidence of a positive radial pressure gradient. This is most noticeable at the 90% height location where the general static pressure level is significantly higher than for the two lower radial positions.

iii) Radial variation of velocity will also have an effect since the pressure differential between the blade surfaces is a function of the flow dynamic head, although the relationship is weakened by the tendency for the mid-span pressure gradient to be imposed on the near-wall fluid. From Fig.4.2.7a it can be seen that the velocity at 90% height over the first 50% or so of the blade chord is rather low, and this is evidenced in Fig.4.2.15b by the somewhat smaller difference in pressure between blade surfaces than for the two other radial locations.

Also from Fig.4.2.7a it can be seen that over the last 20% or so of the blade chord, the velocity at 10% height is decreased. However, since the pressure differential across the passage is only small at this chordwise location, it is difficult to pick out any difference due to this velocity variation.

In order to compare the blade surface pressure measurements with the static pressures derived from the inter-blade five-hole probe, the probe data was extrapolated (linearly) to the blade surfaces. The comparison is shown in Fig.4.2.15c. Agreement between the two sets of data is reasonable on the whole, the only region of significant discrepancy occurring over the first 50% of the suction surface. The extrapolated probe results show significantly higher pressures in this region followed by a flat or falling gradient in the 40% - 50% chord location. These data suggest that suction surface separation bubbles exist; however previous wool tuft tests have clearly shown an absence of any separation. It therefore appears that this anomaly is due to inaccuracies introduced by the extrapolation from the actual data points (plus uncertainties with regard to probe 'wall effects'). This is confirmed by examination of the static pressure at each of the data points nearest to the suction surface which show a fairly uniform change of pressure with blade chord.

Turning now to the blade surface static pressure distribution for the chevron blades, Fig. 4.2.15a, rather than work through the data in a similar manner to the discussion of Fig. 4.2.15b, the differences between the two blade types will be concentrated upon. The first
point to note is that there is no data for the 10% chord position, as a result of the pressure tubing for these tappings having been severed by contact with the lower surface of the rotor disc. It is immediately apparent that there is a significant radial variation of blade surface pressure on both blade surfaces, but particularly so on the suction surface. On both the pressure and suction surfaces it is the pressure at mid-height which is dramatically different to the corresponding data for the straight blades of Fig.4.2.15b. The effect on the mid-height pressures for the chevron blade is to decrease the surface pressure over the bulk of the blade chord. Near the trailing edge the pressures at all points are restored to approximately the same level as at the trailing edge for the straight blades, which has an undesirable consequence for the mid-height suction surface distribution since a greater amount of diffusion takes place in this region. In fact there is some suggestion of a separation bubble in the 'step' of the pressure distribution. Even if no separation bubble were present in this suction surface chevron corner, the boundary layer will experience rapid growth due to the high adverse pressure gradient.

The mechanism by which radial pressure gradients are set up in blade passages where the blades have some dihedral is described by Wang Xhongqi et. al. [76] and can be summarised as follows;

i) The force exerted by the blade on the fluid within the passage is in a direction perpendicular to the spanwise direction of the blade. This is illustrated for a conventional straight blade in Fig.4.2.16a, and the force is seen to be tangential to the pitchwise direction.

ii) For a blade with dihedral, the spanwise direction is no longer perpendicular to the pitchwise direction and therefore the force exerted by the blade on the fluid has a radial component (Fig.4.2.16b). Since the fluid cannot move radially in bulk since it is constrained by the casings, then an opposing force must exist to counteract the radial component of the blade force. This opposing force is provided by a radial pressure gradient which is then superimposed on the pitchwise pressure gradient as indicated in Fig.4.2.16b.

iii) A chevron blade can be described simply as being a blade which incorporates double dihedral. Therefore, using the above argument the pressure distribution in a chevron blade passage is that shown in Fig.4.2.16c.

The reason why the pitchwise pressure gradient at mid-height for the chevron blades is higher than nearer to the walls is unclear, but it is encouraging to note that Shi Jing et. al. [75] observed a similar trend on a curvilinear blade.
Results and Discussion

4.2.4 Flow Conditions at OGV Exit

Fig.4.2.17 shows the velocity profiles for both chevron and straight blades. The profile for the straight blades is that of a well sheared and somewhat casing-biased flow. The bias of the mass flow toward the outer wall agrees with the results from the interblade traverses, as indicated by Fig.4.2.7. The interblade results also showed that this casing bias increased progressively through the blade passage, and therefore the velocity profile at outlet was significantly different from that at inlet. This difference is shown to a greater extent by comparison of the velocity profile at Station 5 (Fig.4.2.17) with that at Station 4 (Fig.4.2.3). This comparison shows a remarkable increase of mass flow near to the outer wall, though it has already been shown that a large part of this flow redistribution takes place prior to the stator leading edge as a consequence of intense mixing promoted by the presence of tip leakage fluid.

Fig.4.2.17 shows that the velocity profile for the chevron blade is characterised by a velocity deficit at mid-height which is associated with the corner or bend of the chevron shape. The deficit at mid-height is balanced by increases in velocity, and hence mass flow, near to the walls.

The contours of axial velocity shown in Fig.4.2.18 provide more detailed information on the velocity variation. Comparison of these two plots with those for the 'low stagger' tests shows that the angle of sector plotted is increased, which is a reflection of the increased spacing for the 'high stagger' blades. Immediately identifiable are the positions of the wakes, and these clearly show the chevron shape of the blade in Fig.4.2.18a, and the straight blade in Fig.4.2.18b.

Fig.4.2.18b shows no evidence of the flow stalling over any part of the blade and therefore confirms the measurements and flow visualisation tests of Section 4.2.3, which showed the blade surface boundary layers to be unseparated. It is interesting to note how the straight-blade wake becomes more pronounced in the regions extending in about 25% annulus height from each wall, in comparison to the wake at mid-height. This wake thickening seems to be the result of secondary flows which develop in the latter part of the blade passage, as discussed in Section 4.2.3, and tend to sweep the low energy wall boundary layer toward, and part way along, the suction surface of the blade. The effect appears to be more pronounced near the outer wall, where between the blade wakes very high velocities persist close to the wall. It is difficult to see from the blade passage flow vector plots of Fig.4.2.12 whether this outer wall secondary flow is stronger than that at the inner wall because the data points do not come close enough to the walls for these near-wall flows to be well enough defined. Near to the inner wall the low energy wall fluid is still largely associated with the end-wall, although there is some evidence of high velocities penetrating into the pressure surface corner, and accumulation of lower energy fluid in the suction surface corner and
within the suction surface wake; both of which are indicative of some secondary flow taking place even though it might be of a reduced strength when compared to the outer wall flow. A further feature noticeable from Fig.4.2.18b is the radial location of the regions of maximum velocity, which cause the casing bias of mass flow seen in Fig.4.2.17.

Many of the above comments concerning the straight blades can also be made for the chevron blades (Fig.4.2.18a). There are some differences though, the most significant of which (apart from the overall wake shape) is the increased wake prominence at mid-height. This part of the wake is both wider and deeper than the corresponding portion of the wake in the case of the straight blade. In the discussion of this same feature in the 'low stagger' tests, it was suggested that this velocity deficit is caused by the two suction surface boundary layers coalescing. However, whilst this may well be a contributory factor, a further likely cause of this enlarged wake is the rapid growth of the suction surface boundary layer within the chevron corner due to the very steep pressure gradient present there (Fig.4.2.15a).

It is noticeable also that in several regions there are minor improvements for the chevron blade in terms of reduced velocity deficit. These regions include; the areas close to the end-walls between the blade wakes, and the blade wakes extending from the walls to about 25% annulus height from each of those walls. These observations are also supported by Fig.4.2.19 which shows that the tangential blockage is greater for the chevron blades in the mid-span region, but either side of this locality then it is lower.

The plots of the flow vectors at all data points are given in Fig.4.2.20. In Fig.4.2.20b, for the straight blades, perhaps the most noticeable feature, aside from the wake position denoted by the abrupt change in direction of the vectors, is the radial component of the velocity which increases toward the outer wall. This trend was also observed for the 'low stagger' blades (Fig.4.1.9) and is due to the outward cant of the diffuser. Fig.4.2.21 shows the same data after this radial velocity component due to diffuser curvature has been removed from each data point.

In Section 4.2.3 it was suggested that classical curved duct secondary flows were being set up in the latter part of the blade passage, but it was not easy to verify this because of a shortage of data points close to the walls. However, in Fig.4.2.20b that such secondary flows are present at exit from the blade is quite clear both at inner wall and outer wall. These flows are characterised by overturning adjacent to the walls and underturning a short distance in from the walls. It is difficult to define the inner edge of the underturning regions since there appears to be a small amount of residual swirl in the outer half of the annulus. The two regions of underturning are however fairly well pronounced, more so than the regions of overturning. This is considered to be a result of the additive effect of the contra-rotating vortices set up in the blade passage as discussed in Section 4.2.3 (see also Fig.4.2.13 which depicts the vortex interaction). At the suction surface/inner wall corner the secondary flow vortex structure is confused, and it is unclear precisely what the flow is doing in this very
localised region.

In order to see how well the inter-blade traverse data agrees with this downstream traverse data, the flow vectors at the 100% chord position of Fig.4.2.12 have been resolved into a plane normal to the axial direction and replotted in Fig.4.2.22. Comparing this data to Fig.4.2.20b shows that the flow underturning in the regions 5-20% and 60-95% of blade height match well. The fluid close to the suction surface in Fig.4.2.22 shows stronger movement toward that same surface than is seen in Fig.4.2.20b since the wake mixing varies strongly between the two axial locations. The reason for the higher pitch angles toward the inner wall for the fluid close to that wall in Fig.4.2.22 is unclear.

The flow vector plot for the chevron blades shows a very similar flow structure to that for the straight blades. The only difference of any significance between the two plots is the change in the wake flow. For the chevron blade there is a tendency noticeable within the wake regions for the fluid to move toward mid-height. The likely reason for this movement is the radial pressure gradients which are set up in the chevron blade passage (Fig.4.2.15a, 4.2.16) and thus produce a low static pressure at blade mid-height.

Fig.4.2.23 shows the pitch-averaged flow yaw angle at this station for the two blade types, the mean level of which is slightly negative thus confirming the suggestion above that there is a small amount of residual swirl in the flow. The mean yaw angle is approximately 2.5° which equates to a blade deviation angle of 10.5°. This value of deviation is the same (within experimental error) as that calculated from the inter-blade traverse data at the 100% chord position (Fig.4.2.14). The reason why the estimated deviation is lower than this value is probably because the estimate is based on two-dimensional cascade correlations, when the actual flow situation is significantly three-dimensional. Fig.4.2.23 clearly shows that the secondary flows near the walls are indeed well developed. The secondary flow near the outer wall spans a greater part of the blade passage than the corresponding flow near the inner wall, although the degree of flow over/underturning is similar. The reasons for this wider spread of the outer wall secondary flow are connected with the way in which the flow develops within the stator blade passage, and are therefore connected with the stator inlet flow conditions. In particular, the highly turbulent tip leakage fluid and the way in which the incidence varies with radius are important.

It is interesting to note that these near-wall secondary flows are stronger for the straight blade than for the chevron blade. This confirms the result of Shi Jing et. al. [75] who have showed that for blades with dihedral the strength of the conventional secondary flows is reduced.

A further point to note from Fig.4.2.23 is that there is a small increase in flow swirl close to mid-height for the chevron blades. This arises due to the distortion of the effective flow passage at this radial location by the rapid growth of the suction surface boundary layer at the chevron corner.
The performance of the two types of 'high stagger' blade is given in quantitative terms in Table 4.3. The highly distorted velocity profile which has been noted at the station prior to the OGV's (Fig.4.2.3) is again indicated by the relatively high kinetic energy flux coefficient. As seen from the above, this distorted stator inlet velocity profile changes rapidly in the distance up to the stator leading edge, as the rotor wake and tip leakage flows mix out. Therefore whilst from Table 4.3 the kinetic energy flux coefficient decreases across the blade row, if the flux coefficient were to be calculated immediately prior to the stator leading edge then it is likely to yield a lower value and hence the change in flux coefficient across the blade row would be significantly smaller. At stator blade outlet the flux coefficient is virtually the same for the two blade types. Considering the presence of the thick mid-height wakes for the chevron blades, it might be expected for the outlet flux coefficient for this blade to be higher, but the effect on $\alpha$ of this larger wake region is balanced by the marginally shallower wakes at other radial locations.

The calculated loss for the straight blade (8.3%) is slightly higher than the predicted loss (Horlock[3]) of 6.1%. The predicted figure is based on cascade correlations and hence can be expected to be slightly lower than achieved in an actual compressor because it does not take account of; losses due to the continuing mixing out of the rotor wake and tip leakage flows, radial variation of velocity, radial variation of incidence. Therefore the loss figure of 8.3% for the straight blades is considered to be a realistic loss level for this blading. The loss for the chevron blades is almost 3% higher than for the straight blades, which is considered to be due to two primary factors; firstly an increase in skin friction loss caused by the increased wetted surface area of the blade, and secondly increased mixing loss from the large blade wake at mid-height.

The way in which the blade losses are distributed over the annulus are shown in Fig.4.2.24 which presents contours of constant loss level. The distribution of Fig.4.2.24b for the straight blades shows that the bulk of the loss is associated with the blade wake and the inner wall boundary layer. Near to the outer wall, especially in the region close to the pressure surface, a negative loss is evident. The reason for this marked difference between the inner and outer wall boundary layers is the way in which mass flow tends to migrate outwards as it progresses through the blade passage, as seen in Fig.4.2.7. By this means the fluid energy level near the outer wall is increased by higher energy fluid moving from the main part of the blade passage, whilst near the inner wall the energy level of the fluid is reduced by the radial migration. It will also be noticed from Fig.4.2.24b that in the near-wall regions the highest loss occurs toward the suction side of the blade passage, which is the result of the movement of the boundary layer fluid under the influence of the blade passage secondary flows. The loss associated with the stator blade wake is fairly uniform over the major part of the annulus height, although in the region from approximately 65-95% annulus height a significantly larger contribution to the overall loss is observed. This high loss region is the result of the
normal level of wake loss being increased by the presence of low energy fluid swept from the outer wall region by strong secondary flow action. The reason why such a feature does not also appear in the wake near the inner wall is perhaps due to the secondary flow in this region being confined to a smaller part of the blade span and the low energy fluid being contained within the secondary flow region rather than moving along the blade suction surface. The last feature to note from this Figure is the low level of positive loss which is present in the main part of the flow passage, which is thought to be due to the continuing mixing out of rotor blade wakes as they pass through the stator blade passages.

The loss distribution for the chevron blades shows many similar features to those discussed above for the straight blades. However, there are notable differences in the loss associated with the stator blade wake. Apart from the obvious point that the shape of the chevron is reflected in the wake pattern, there is a larger region of loss at mid-height within the wake. This region corresponds to the velocity deficit noted from the velocity contour plot of Fig. 4.2.18. It is interesting to note that this high loss region seems to have 'joined up' to the high loss region in the wake at the outer radii, thus creating a high loss wake from approximately 25-90% annulus height. It is this area which is a primary cause of the higher loss noted for the chevron blades from Table 4.3.
Results and Discussion

4.3 'Rainbow' Compressor Test Results

4.3.1 Proving the 'Rainbow' Technique

Ideally, to test a modified stator blade of some description, a completely new stator ring should be manufactured and fitted in order to establish the performance of the new blade type. However, such a procedure is very time consuming and costly, particularly when it is required to test a large number of modifications, some of which may only be quite minor. Therefore it was decided to attempt testing using a 'rainbow' technique whereby only a small number of the blades within a stator blade ring would be changed for the modified type of blade. This type of testing is obviously advantageous when a number of blade modifications are required to be tested in order to reveal those blade types which show promise of performance improvements. Should such promise be shown in a blade type, then it was envisaged that the next step in the testing would be the construction and testing of a complete blade ring. Therefore this 'rainbow' technique is to be viewed as a preliminary to definitive blade testing.

In order to establish the validity of the test technique a control experiment was performed. To do this the complete ring of 'high stagger' straight blades was modified to make a sector of the ring removable. This sector covered nine blade spaces, which equates to approximately 10% of the blade ring since the total number of stator blades is 92. Nine chevron stator blades were put into this section for the control experiment, since results of tests on this blade configuration could be compared with the results from a complete ring of chevron blades reported in Section 4.2.

A traverse performed upstream of the 'rainbow' sector revealed the flow directions and velocity distributions to be the same within experimental error as measured for the complete blade ring. However the mean level of static and total pressure were found to be different, thus indicating the need for an upstream traverse to be performed if blade performance figures such as pressure loss and recovery are required.

Downstream of the 'rainbow' section the pitch-averaged velocity distribution is shown in Fig.4.3.1 where it is compared with the result for the complete chevron ring. Agreement between the two curves is thought to be sufficient for the purposes for which this test technique is to be used. Further detail of the downstream velocity distribution are shown by the velocity contours of Fig.4.3.2, which indicates that the distribution is very similar to that for the complete chevron ring (Fig.4.2.18a).

The quantitative performance is detailed in Table 4.3 and shows that the calculated blade loss is the same as for the complete chevron ring within experimental error. On the basis of both the qualitative and quantitative results discussed above, it seems that the 'rainbow' technique is quite suitable for preliminary test work on blading.
4.3.2 Comparison With Tests Due to Harasgama

It has already been pointed out in Section 4.1.3 that the significant improvement in downstream velocity distribution for chevron blades seen by Harasgama was not a feature of the 'low stagger' tests. Furthermore in the 'high stagger' tests there was no sign of such an improvement either. Therefore possible reasons for this difference were explored, but before this could be done the nature of the change in velocity distribution was isolated. From Fig.4.3.3b it can be seen that in Harasgama's tests of a straight blade that a poor velocity distribution near the inner wall was caused by a suction surface hub corner stall. Moving on then to his chevron blade test (Fig.4.3.3a), this corner stall region can be seen to have been eliminated, and it is this change which is the cause of the velocity profile improvement seen by Harasgama. In both 'low stagger' and 'high stagger' tests reported here there was no hub corner stall region observed for the straight blades, and therefore no chance of improving the velocity profile using chevron blades in a similar manner to Harasgama. The question then arises as to why did Harasgama measure a hub corner stall with his straight blades when with similar blading ('low stagger') it was not observed here.

The first possibility considered was that the difference could be due to a different blade chord Reynolds number between the tests. It was thought that the higher Reynolds number used in the tests reported here could be the cause of the hub corner boundary layer remaining attached. In order to test this suggestion, the rig was simply run at a lower speed than before (500rpm compared with 2000rpm for the tests reported in Section 4.2) such that the blade chord Reynolds number became comparable to that used by Harasgama (0.4x10^5). An area traverse was performed downstream of the blade row with the rig operating at this low speed, the result of which is shown in the velocity contour plot of Fig.4.3.4. The contours are seen to be very ragged as the transducer resolution is approaching its limits at the low dynamic head involved, and also there are a number of points out off calibration as evidenced by the blank areas in the wake region. However, the flow pattern seems to be essentially the same as reported before in Fig.4.2.18b, with no sign of a large hub corner stall. Therefore the possibility of Reynolds number influence being the cause of the hub stall observed by Harasgama must be discounted. The influence of the Reynolds number change is limited to the blade wakes which are somewhat more severe in Fig.4.3.4. This confirms that the Reynolds number for the bulk of the tests reported here is sufficient to be well clear of the critical Reynolds number range.

The second avenue of investigation was concerned with the hub geometry of the stator blade. The stator blades used in the current tests are completely shrouded whereas those used by Harasgama incorporated a small hub clearance in order to necessitate rotation of the OGV's. In trying to minimise the hub clearance Harasgama had employed felt pads between each blade and the inner casing, but was unable to form a properly sealed shroud. To
Results and Discussion

reproduce this hub clearance a 'rainbow' section of blades was modified to give a hub clearance of approximately 0.25mm, or 0.4% of the annulus height. The results of the test performed are shown in Fig's 4.3.5, 4.3.6, 4.3.7. These figures clearly show that the hub corner stall is now present and it causes the pitch-averaged velocity profile to be poor near to the inner wall.

Therefore, it has been ascertained that the reason why Harasgama observed dramatic improvement in blade outlet velocity profile for the chevron blades is due to the elimination of a hub corner stall which in turn arose from the use of blades with hub clearance. This result is valuable since there are situations where hub clearance is used on some engines, and in such situations the use of stator blades with dihedral at the hub will help to eliminate large scale passage blockage due to leakage flows. Han Wanjin et. al. [77] confirm this by reporting that an obtuse wall-suction surface corner angle serves to suppress a hub stall.

The calculated stagnation pressure loss for this hub clearance blade is approximately 4% higher than for a shrouded blade (Table 4.3). That the blade loss increases when hub clearance is employed is intuitive from Fig.4.3.6 which strongly suggests a higher loss with the region of separated fluid in the hub corner.
4.4 Diffuser Test Results

4.4.1 Diffuser Performance With Four Inlet Conditions

The four inlet conditions under which the diffuser has been operated are differentiated by the form of compressor stator blading used i.e. chevron-shaped and straight blades in both 'low stagger' and 'high stagger' form. The variation in pitch-averaged velocity profile at diffuser inlet for these four conditions is shown in Fig.4.4.1, which indicates well-sheared profiles with thick boundary layers. Other features to note from this Figure are; the trend for the profiles for the 'high stagger' blades to be relatively casing-biased, and the mid-height velocity deficit for both chevron blades. The amount of inlet swirl also varies between the four test runs. Fig.4.4.2 shows that the two 'high stagger' configurations both give a small average flow swirl of about 2.5°. For the 'low stagger' blades there is a significant difference in flow swirl; being a mean of approximately 0° for the straight blades but for the chevron blades it varies from about 2° for the inner half of the annulus whilst peaking at about 9° in the outer annulus half.

As has already been indicated in Chapter 1, the amount of flow swirl in a diffuser has an influence on the radial distribution of flow within the diffuser (Horlock [59], Elkersh et. al. [100], Lohmann et. al. [101]). Fig.4.4.3 shows the pitch-averaged velocity profiles at diffuser exit and comparison of the two 'low stagger' curves confirms the expected radially outward movement of flow for the 'low stagger' chevron blades for which the highest inlet swirl exists. The reason why the two curves for the 'high stagger' blades should be casing biased is not connected with flow swirl but is rather a reflection of a similar casing bias at diffuser inlet (Fig.4.4.1).

The way in which the velocity profile develops along the diffuser length is revealed in Fig.4.4.4. It is noticeable for all four inlet conditions that a greater degree of change takes place in the inner wall layer than the outer wall layer (although for the 'low stagger' straight case this trend is not so marked). The development of the wall boundary layers is detailed further in the distributions of boundary layer parameters of Fig's 4.4.5, 4.4.6. Looking firstly at the parameters for the inner wall boundary layer (Fig.4.4.5), all three parameters are seen to increase steadily as would be expected for a diffusing flow passage. For both chevron and straight blade types the displacement thickness and momentum thickness are greater for the 'high stagger' blades than the 'low stagger'. The shape parameters show that the chevron blades give an inner wall boundary layer with a lower value and hence a reduced tendency to separation; apart from the last point for the 'low stagger' chevron case which is higher than for the other three cases. This sudden change in boundary layer is also seen in the velocity profiles of Fig.4.4.4a though the reason behind the sudden change is unclear.

The boundary layer parameters for the outer annulus flow show a somewhat different
picture. Fig.4.4.6 indicates a large variation between the curves in terms of boundary layer thickness and growth rate. At every point down the diffuser the parameters show lower values for the 'high stagger' blades. It is also interesting to note the lower parameter values for the chevron blades when compared to the straight, in both the 'low stagger' and 'high stagger' configurations. Apart from the 'low stagger' straight case there is little change in the shape parameter for this outer wall flow along the diffuser length which contrasts with the inner wall flow where a progressive increase is seen (Fig.4.4.5c). This reflects the casing bias of the flow for these three inlet conditions. The adverse pressure gradient has a large effect on low momentum fluid regions, and since the inner wall flow is of lower momentum than the outer layer at diffuser inlet, it is therefore the inner wall flow which suffers the most retardation and hence boundary layer growth. This then creates additional blockage near the inner wall, causing fluid to be moved outwards and thus maintaining the outer wall boundary layer in good shape.

Comparing Fig.4.4.5c & 4.4.6c with the data due to Harasgama [81] reveals substantial differences in the shape parameter for the flows on both walls. For the inner wall flow Harasgama reports a major improvement for chevron blades for which his shape parameter stays almost constant over the bulk of the diffuser at approximately 1.6 whilst for his straight blades a progressive increase up to about 2.7 at diffuser exit is indicated. This major change in wall boundary layer behaviour is very likely to be connected with the difference in inlet velocity profile caused by hub clearance effects as discussed in Section 4.3. For his outer wall flow Harasgama shows large fluctuations in the boundary layer shape parameter though the general trend is still one of improvement when chevron blades are used at diffuser inlet.

For a diffuser with outward curvature (such as that used here), Stevens [95] indicates that for fully developed inflow it is normal for the outer wall boundary layer to grow more rapidly than the inner wall layer as a result of the steeper pressure gradient on the outer wall which arises due to the flow curvature. The fact that for Harasgama's tests [81] and the tests reported here, the opposite effect is seen with the inner wall layer having the higher growth rate and shape parameter, is indicative of the major influence of the diffuser inlet flow distortions which arise with a compressor sited immediately prior to diffuser entry.

The compressor-related flow distortions not only take the form of radial redistribution of fluid but also circumferential asymmetries due to stator blade wakes. These blade wakes and evidence of their decay is given by Fig's 4.4.7 - 4.4.10 which show axial velocity contours at several stations down the diffuser for all four inlet conditions. A further feature which can be deduced from these plots is whether significant flow swirl is present since the blade wakes will move circumferentially according to the degree of swirl. Fig.4.4.2 indicates a high level of flow swirl in the outer annulus half for the 'low stagger' chevron blades. This is confirmed in Fig.4.4.7 where the wake in the outer annulus half is seen to move circumferentially whilst the wake in the inner annulus half remains in nominally the same circumferential position.
Results and Discussion

Fig.4.4.10 exhibits a circumferential wake movement near both walls for the 'high stagger' straight case which correspond to the under-turning regions evident in Fig.4.4.2, though it is of reduced strength compared with the outer wall flow for the 'low stagger' chevron case.

A further way of indicating the behaviour of the blade wakes is by the distributions of Fig's 4.4.11 & 4.4.12 in which the depth of the blade wakes (i.e. the velocity change from peak to trough) is portrayed. Fig.4.4.11 shows several features:

i) the general trend for the wake depth to decrease along the diffuser length

ii) the greater wake prominence in the regions close to the walls, which is particularly noticeable for the 'low stagger' blades

iii) the deep wake associated with the mid-height corner of the chevron blade in the early stages of the diffuser. This wake region is also clearly seen in the contour plots of Fig's 4.4.7, 4.4.9

iv) the increased wake depth at the first position within the diffuser for the 'high stagger' blades when compared with the 'low stagger'. This increase in wake depth is a reflection of the greater chord length (see Appendix 1) of the 'high stagger' blades.

The behaviour of the blade wakes in the axial direction is particularly important since in order for the combustor to work satisfactorily with no hot-spots in the circumferential direction then the feed air to the various primary and dilution holes must be circumferentially uniform (i.e. free from blade wakes). Therefore there should be little or no evidence of wakes at diffuser exit. Stevens et. al. [103] have indicated that blade wakes in a parallel duct decay within two blade chord lengths to a very low level ($1 - u_\text{u} / u_\text{z} = 0.1$). In an adverse pressure gradient however wake decay will be slower, and if the gradient is sufficiently severe then the wakes may grow rather than decay (Hill et. al [107]).

In the tests reported here the diffuser is of conservative design and therefore the pressure gradient is modest. As a result of this the wakes are seen to decay within the diffuser to an acceptable level at diffuser exit ($1 - u_\text{u} / u_\text{z} = 0.1$). It is interesting to note that for both the 'low stagger' tests the wake decay rate is initially slower near the walls than the centre span region, which thus gives rise to noticeably more significant wakes in these regions all the way to diffuser exit. It is encouraging to note that the mid-height velocity deficit for the chevron blades mixes out well such that by diffuser exit it is no more prominent than the wakes over the rest of the span.

Fig.4.4.12 should be viewed in conjunction with Fig's 4.4.7 - 4.4.10, which indicate that towards diffuser exit the wakes have fully mixed out in most regions. The reason why Fig.4.4.12 still shows a finite wake depth in these locations is simply due to the 'noisy' nature of the velocity data as evidenced by the ragged contours at diffuser exit in Fig's 4.4.7 - 4.4.10.

Fig's 4.4.12a & 4.4.12b can be compared directly with that of Harasgama [81] (Fig.4.4.13) since both the blading and diffuser are similar. The comparison shows that
Whilst the calculated wake depth at diffuser exit is similar for both sets of tests, the decay within the diffuser is generally slower for Harasgama’s tests than for the tests reported here. The probable reason for this is that the slower decay in Harasgama’s diffuser is caused by a higher diffuser area ratio (2.0 c.f. 1.685 for the tests reported here) and hence a more adverse pressure gradient.

Fig.4.4.14a shows the diffuser pressure recovery for all four inlet conditions as calculated from the mass-weighted mean static pressures. Whilst the two curves for the ’low stagger’ blades show good agreement and a progressive increase in pressure, the curves for the ’high stagger’ blades show less agreement and the data seems quite ‘noisy’. Fig.4.4.14b shows a similar plot except that it indicates pressure data measured from outer wall tappings. Unfortunately no measurements of the diffuser wall static pressure tappings were recorded for the test run with ’high stagger’ chevron blades. However, the two curves for the ’low stagger’ tests show reasonable agreement and also tie in well with the data of Fig.4.4.14a. The curve for the ’high stagger’ straight blades of Fig.4.4.14b shows a progressive increase in pressure along the diffuser but does not agree well with the data of Fig.4.4.14a. One reason why the data for the ’high stagger’ tests shows a greater degree of uncertainty is due to the lower axial velocity (approximately 75% of that for the ’low stagger’ tests) which means the axial dynamic head will be about 56% of that for the ’low stagger’ tests and hence the transducer resolution becomes worse (i.e. the signal to ’noise’ ratio is nearly halved).

Table 4.2 shows the overall diffuser performance data as well as that for stations within the diffuser. Looking firstly at the overall diffuser performance presented in Table 4.2a it is clear that within both configurations of blade stagger the diffusers with chevron blades at inlet out-perform those with straight blades at inlet, in terms of higher pressure recovery (and hence also higher diffuser effectiveness), lower loss and reduced exit kinetic energy flux coefficient. It is particularly encouraging to note for the ’high stagger’ chevron configuration that the kinetic energy flux coefficient is actually lower at diffuser exit than it is at inlet. Reduction of the kinetic energy flux at diffuser exit is important since it means that the dump loss is reduced if the diffuser exits into a plenum or combustor region at this plane. A further beneficial effect is connected with the flow around a downstream combustor head. It has been shown (Stevens & Wray [120]) that large accelerations of the flow take place over the combustor head. If the flow at exit from the diffuser has a less peaked velocity profile it means that there is more fluid already turning away from the mean flow centre-line and hence the flow accelerations around the combustor head will be reduced and therefore the associated losses will also be reduced.

Harasgama’s tests (Table 4.2a) also show improved diffuser performance for chevron blades at inlet compared with straight blades at inlet. However, the very large improvement in exit kinetic energy flux and pressure recovery is in part due to the elimination of a hub corner stall with the chevron blades and hence an improved velocity distribution near the inner wall at diffuser inlet. With straight blades at inlet Harasgama reported a very large increase in
boundary layer thickness which caused the exit kinetic energy flux coefficient to be high, and therefore the pressure recovery and diffuser effectiveness to be low. The diffuser used by Harasgama was of a slightly larger area ratio than the one used for these tests (2.0 c.f. 1.685) which is probably the reason for the higher loss and lower diffuser effectiveness when compared with the similar 'low stagger' tests reported here.

Stevens [95] has tested the same diffuser as that used by Harasgama, except that he used fully developed inflow. In his results Stevens has shown that the outer wall boundary layer thickness increases greatly (in comparison with his inner wall layer) and it is this which causes the very high exit kinetic energy flux coefficient and hence the low diffuser effectiveness.

Table 4.2b indicates the diffuser performance along its length. One noticeable feature is that whilst a general trend exists for the kinetic energy flux coefficient to increase over the diffuser length (due to growing wall boundary layers), for the 'low stagger' straight test and both the 'high stagger' tests the coefficient decreases initially. This decrease is the result of the blade wakes mixing out since the coefficient is not based on the pitch-averaged velocity profile but rather on the data covering one complete blade space.

The losses of Table 4.2b also deserve some comment. It is noticeable that for all four configurations that the calculated loss is negative for part of the diffuser length. This is clearly not truly representative since it implies energy addition to the diffuser flow, which is not the case. Some shortcoming of the mass flow correction procedure can also be discounted as a cause of these negative losses since there is no correlation between the mass flow discrepancy and the loss in Table 4.2b. Rather it seems that the most likely cause of the discrepancy is the inability of the five-hole probe technique to accurately deal with strong blade wakes, since the traverse station at which the diffuser inlet reference conditions are calculated is that station where the stator wakes are most prominent. It is difficult to be more precise in accounting for this discrepancy without doing some accurate control experiments using the five-hole probe in distorted flows.

4.4.2 Diffuser Reynolds Shear Stress

It has been previously shown (Stevens & Williams [102]) that increased turbulent mixing has the effect of reducing the diffuser exit kinetic energy flux coefficient. Subsequently Stevens et. al. [103,105] suggested that an increase in turbulent mixing within the diffuser could be achieved by positioning the compressor very close to diffuser inlet so that blade wakes and highly turbulent fluid are present. However, they did not indicate the actual level of turbulent shear stress within the flow.

In a detailed investigation of diffuser flows, Stevens [95] made measurements of the velocities and turbulence structure at several stations within the diffuser when operating with
fully developed inlet flow. He showed that the measured shear stress distributions exhibit considerable lag behind the velocity profile development (also reported by others e.g. Williams [121]). This lag means that the shear stress distribution at diffuser exit should still reflect something of the inlet conditions. In particular, if the compressor is positioned close to diffuser inlet then the increased turbulent mixing within the diffuser should be reflected in higher levels of turbulent shear stress at exit.

Using the same diffuser as that used by Stevens [95], Harasgama [81] made similar measurements of the turbulence structure, this time with a compressor sited at diffuser inlet. However, some doubt was cast on his results since they indicated lower levels (approximately half) of diffuser exit Reynolds stress when compared to Stevens result. Therefore a need still exists for reliable diffuser exit Reynolds stress data, when the diffuser is operating with a compressor at inlet.

The tests reported here involve the use of such a compressor/diffuser combination, and whilst the diffuser area ratio (1.685) is a little lower than that used by Stevens [95] and Harasgama [81] (2.0), the non-dimensional length is the same \((L/\Delta r_{\text{inlet}} = 5.0)\). However, due to time constraints it was not possible to make turbulence measurements on this test rig. Nevertheless an estimate of the diffuser Reynolds shear stress distributions has been made using mean velocity data and a simplified form of momentum equation. Reynolds [122] gives the streamwise mean momentum equation for the boundary layer as

\[
\frac{u}{\partial x} + \frac{v}{\partial y} = \frac{1}{\rho} \frac{\partial p}{\partial x} + u \frac{\partial^2 u}{\partial y^2} - \frac{\partial (u'v')}{\partial y} - \frac{\partial (u'^2)}{\partial x} \tag{4.4.1}
\]

This equation can be further simplified by the elimination of the 2nd and 4th terms on the r.h.s. of the equation. The 2nd term on the r.h.s. is a laminar shear stress term and since the flow is turbulent, this only has any significance in a very small layer close to the wall and therefore is ignored. The 4th term is the Reynolds normal stress, which is known to be small compared to the Reynolds shear stress term. Thus Equation 4.4.1 becomes

\[
\frac{u}{\partial x} + \frac{v}{\partial y} = \frac{1}{\rho} \frac{\partial p}{\partial x} - \frac{\partial (u'v')}{\partial y} \tag{4.4.2}
\]

and rearranging gives

\[
\frac{\partial (u'v')}{\partial y} = \frac{1}{\rho} \frac{\partial p}{\partial x} - u \frac{\partial u}{\partial x} - v \frac{\partial u}{\partial y} \tag{4.4.3}
\]

Equation 4.4.3 is the equation used here to estimate the diffuser Reynolds shear stress
distribution. Because data from one traverse station does not give any indication of the axial variation of velocity and pressure, therefore data from two adjacent traverse stations is required, the data used being the pitch-averaged distributions of axial velocity and pressure. In order to get values of \( v \), the 2-d continuity equation is used

\[
\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = 0
\]

This method involves several assumptions:

i) The duct is assumed to be two-dimensional. This will only incur small inaccuracies since the inlet radius ratio is very high (0.88).

ii) Both the mean and fluctuating components of circumferential velocity are assumed to be zero. Whilst this assumption is reasonable close to diffuser exit, clearly it becomes less so toward diffuser inlet where e.g. blade wakes and secondary flows are still present.

iii) The flow is assumed to be axisymmetric. Again this will be more valid close to diffuser exit than at the inlet.

The calculations have been carried out at four positions down the diffuser, these positions being mid-way between traverse data from stations W5, W6, W8, W10, W12. The calculations have not been reported for the 'high stagger' test since the relatively low axial velocities meant that insufficient transducer resolution was available to give adequately accurate velocities and pressures for the calculation to be meaningful.

The calculation was performed in the following fashion in which the main features are indicated:

i) The pitch-averaged distributions of axial velocity and static pressure are calculated for the two traverses.

ii) Beginning at the inner wall, calculations are performed at every 1% annulus height as follows in steps iii) to x).

iii) \( \partial u/\partial y \) is calculated at that point for each traverse, and then averaged.

iv) \( \partial u/\partial x \) is calculated, and thus \( \partial v/\partial y \) from Eqn.4.4.4. When mid-annulus height has been reached, the sign of \( \partial v/\partial y \) is changed in order to ensure that \( v \) tends toward zero as the outer wall is approached.

v) \( v \) is calculated from a running integration of \( \partial v/\partial y \), the boundary condition being \( v=0 \) at the inner wall.

vi) \( v.\partial u/\partial y \) is calculated.

vii) the average "u" is calculated, and thus "u.\partial u/\partial x".

viii) \( 1/\rho \cdot \partial p/\partial x \) is calculated.

ix) \( \partial(u'v')/\partial y \) is thus calculated.

x) \( \partial(u'v')/\partial y \) is integrated and then non-dimensionalised to give \( (u'v')/\partial^2 \), using the condition of \( (u'v')=0 \) at the inner wall.

Fig.4.4.15 indicates the form of Reynolds stress distribution which is expected through
the diffuser. The data is that of Stevens [95], and is for a diffuser with fully developed inflow.

The results of the calculations for the 'low stagger' tests are given in Fig. 4.4.16. Comparing with Fig. 4.4.15 it is clear that the results closest to diffuser inlet have been strongly affected by the relatively high three-dimensional nature of the flow. As was indicated earlier, a poor result at diffuser inlet is to be expected because of the three-dimensional nature of the flow.

For the three curves of Fig. 4.4.16 closest to diffuser exit the comparison with Fig. 4.4.15 holds up rather better. Comparison of Fig. 4.4.16 with the pitch-averaged velocity profiles of Fig. 4.4.15 shows that the radial location at which the Reynolds shear stress changes sign does not coincide with the point of maximum velocity. Bradshaw [123] indicates that in an asymmetric flow there is no reason why the point of zero net Reynolds shear stress should coincide with the point of zero velocity gradient, since the Reynolds stress can be transported normal to the flow direction by the turbulence itself. However, the difference in radial location for these tests is thought to be too large to be wholly accounted for by transport of Reynolds stress; the other factor causing the difference being some inaccuracy in the calculation.

The values for the three terms on the r.h.s. of Eqn. 4.4.3 are indicated in Fig’s 4.4.17 & 4.4.18 for all the positions at which the calculations have been carried out. It is apparent from examination of these figures that it is the 1st and 2nd terms on the r.h.s. of Eqn. 4.4.3 which have the major influence. The third term is consistently of a lesser significance.

Comparing the actual levels of turbulent shear stress with those of Stevens [95], Table 4.5 shows an encouraging trend in the expected direction i.e. that the maximum values of Reynolds shear stress in both the inner and outer layers of the flow are higher than those in Stevens test. Stevens & Nayak [124] have performed a similar prediction of the Reynolds stress in a diffuser, but have also compared their results with experimental test data. They found that at diffuser exit the maximum level of Reynolds shear stress was under-predicted by approximately 40%. It is therefore likely that the Reynolds stresses of Fig. 4.4.16 are also lower than the actual levels. This seems to confirm the initial supposition that the mechanism by which the positioning of the compressor close to diffuser inlet causes improvements in exit velocity profile is by increased turbulent mixing.

The calculation technique involved here is obviously not as accurate as actually making measurements. Since the calculations are performed in one direction across the annulus from inner to outer wall, then the value of turbulent shear stress nearest to the outer wall indicates something of the reliability of the result and the integrity of the data and calculation technique. The turbulent shear stress should have a value of zero at each wall; the inner wall value is fixed at zero since it is the start point of the calculation, but the value at the outer wall will only come close to zero if the data and calculation technique are reasonable.

An attempt was made to improve the accuracy of the result at the diffuser exit location by using a pressure gradient value based on diffuser wall static pressure tappings rather than the
Results and Discussion

pressure gradient as calculated from the five-hole probe results. However, whilst for the 'low stagger' straight blade run this produced almost identical shear stress results, for the 'low stagger' chevron blade test some discrepancy was noted in the shear stress distribution, particularly at the outer wall between the last data point and the condition of zero turbulent shear stress at the wall. This result indicates that (at this diffuser exit location) because the terms on the r.h.s. of Eqn 4.4.3 are small (Fig's 4.4.17, 4.4.18) the calculation of Reynolds stress becomes sensitive to the small variations of measured flow quantities which arise due to the limitations of the instrumentation. Therefore there still exists a need for accurate turbulence measurements in this type of diffuser configuration.
Results and Discussion

4.5 Overall OGV/Diffuser Performance

Table 4.4 shows the performance figures for the complete OGV/diffuser system for both 'high stagger' and 'low stagger' and also includes performance data due to Harasgama[81]. As has already been observed in Section 4.4.1 the diffuser exit kinetic energy flux coefficient is improved (i.e. reduced) for 'low stagger' chevron blades, 'high stagger' chevron blades and also the chevron blades tested by Harasgama, when these are compared to their corresponding straight blade figures. However, whilst these diffuser exit conditions are improved for the chevron blades, as is the isolated diffuser performance (Table 4.2), the OGV performance tends to dominate the overall losses and a less clear trend is evident for the OGV performance (Table 4.3). It has already been shown from Table 4.3 that in the case of the 'low stagger' test the chevron OGV performs best in terms of pressure recovery and loss but the situation is reversed for the 'high stagger' blades. This situation is repeated for the overall performance figures of Table 4.4. The reason why this difference in performance exists between the 'low stagger' and 'high stagger' runs is unclear. It is interesting to note that in Harasgama's test the chevron blade produced an improved overall pressure recovery with nominally the same level of loss when compared to his straight blade.

The improvement in diffuser exit velocity profile therefore must be weighed against the higher overall OGV/diffuser loss. This process of assessing the overall advantages of using chevron blades is complicated, and is in the hands of the engine manufacturers since a large number of auxiliary factors may need to be considered e.g.;

i) improving the diffuser exit velocity profile could improve primary and dilution hole feed to the combustor with beneficial consequences for the turbine stages in terms of reduction of hot-spots,

ii) the compressor outlet guide vanes may be required to transmit structural loads and the use of a chevron type of blade may reduce the load-carrying capability of the blade row.
CHAPTER 5 CONCLUSIONS
Conclusions

A large-scale compressor/diffuser test rig has been designed and constructed which, together with an automated data acquisition system, permits more detailed and more accurate measurements than were previously possible - especially in the region of the compressor OGV’s.

Results are presented of experiments carried out using two different single-stage axial-flow compressors operating immediately upstream of a straight-core annular diffuser, each compressor being tested with a conventional stator row and a double-dihrdral chevron type of stator row i.e. four main configurations have been investigated. In addition to these main tests, the effects of operating the stator row at low Reynolds number, and of operating the stator row with a small hub clearance have been investigated.

The following points relate to the OGV’s in particular:
- The OGV losses calculated from the results are consistently higher than the losses predicted from cascade correlations due to the inability of the cascade correlations to adequately deal with thick inlet boundary layers, radial variation of incidence and the mixing out of wakes from the upstream rotor.
- The chevron type of OGV tends generally to incur a higher loss than the corresponding straight blade, due to increased wetted area and a relatively large wake at mid-height.
- Inter-blade traversing on the ‘high stagger’ straight OGV’s has shown a progressive redistribution of flow throughout the blade passage, brought about largely by the intense mixing associated with rotor tip leakage fluid. These inter-blade results also reveal that the considerable radial variation of incidence onto the OGV’s causes strong secondary flow vortices to be set up in the early part of the blade passage rotating in the opposite sense to classical curved duct secondary flows. This vortex structure has not been reported before.
- A low Reynolds number test on the ‘high stagger’ blades (Re = 0.4x10^5, c.f. 1.8x10^5 for the rest of the ‘high stagger’ tests), showed an increase in wake size. This factor, together with the higher blade loss reported by Harasgama [81] at a similar low Reynolds number (0.4x10^5), implies that the bulk of the tests reported here were performed at a flow speed above the critical Reynolds number range.
- Hub clearance was introduced on a small number of ‘high stagger’ straight blades, which produced a hub corner stall similar to that seen by Harasgama[81].
- For a situation where hub clearance is required on the stator blades, then the incorporation of blade dihedral near the hub will minimise passage blockage due to hub corner stall.

The following points relate to the diffuser and the OGV/diffuser system:
- The positioning of the OGV trailing edge at diffuser inlet causes blade wakes to be present in the diffuser. This causes very little change in the diffuser loss, and significantly improves the diffuser exit velocity profile. As well as reduced kinetic energy flux, there is
also increased Reynolds shear stress at diffuser exit.
- The use of chevron OGV's at diffuser inlet, instead of straight OGV's, brings further improvements in velocity profile at diffuser exit, and a slightly reduced diffuser loss.
- Further improvements in diffuser exit velocity profile seem possible by careful design of blade shape to encourage radial movement of fluid.
Recommendations for Further Work

The base-line straight-blade geometry could be further investigated, by e.g. performing instantaneous flow measurements in order that the contribution to the flow structure caused by fluid shear stresses be assessed.

Also of interest would be the extension of the inter-blade traversing to rig builds with different inlet conditions, which will therefore provide a more comprehensive database on blade passage flows.

Additional designs of blade shape should be tested which encourage radial movement of flow. Stress should be placed on achieving this flow movement in a low-loss manner. The blade design should be tailored to suit particular engine geometries e.g. where the downstream diffuser is outwardly canted then the blade should be designed to encourage radially outward flow, particularly near the outer wall. Inter-blade traversing could be used to good effect on these modified blade shapes to show precisely how the blade passages flows are affected.
References


References


References


References


References


References


References


72 "Rolls-Royce Technology", Rolls-Royce publication, 1986.


References


References


114 Wray, A.P. - PhD thesis, to be submitted.


125 Instruction Manual for ADAC 1012 ADC, ADAC Corporation Europe Ltd.
TABLES
Pressure tappings are located:
i) on both straight and chevron 'high stagger' OGV's
ii) on pressure and suction surfaces
iii) at 10%, 50% and 90% duct height
iv) at 10%, 20%, 30%, 50%, 70% and 90% blade chord

**Pressure side of blade**

<table>
<thead>
<tr>
<th>L/E</th>
<th>O/Wall</th>
<th>I/Wall</th>
</tr>
</thead>
<tbody>
<tr>
<td>P13</td>
<td>P7</td>
<td>P1</td>
</tr>
<tr>
<td>P14</td>
<td>P8</td>
<td>P2</td>
</tr>
<tr>
<td>P15</td>
<td>P9</td>
<td>P3</td>
</tr>
<tr>
<td>P16</td>
<td>P10</td>
<td>P4</td>
</tr>
<tr>
<td>P17</td>
<td>P11</td>
<td>P5</td>
</tr>
<tr>
<td>P18</td>
<td>P12</td>
<td>P6</td>
</tr>
</tbody>
</table>

**Suction side of blade**

<table>
<thead>
<tr>
<th>L/E</th>
<th>I/Wall</th>
<th>O/Wall</th>
</tr>
</thead>
<tbody>
<tr>
<td>S13</td>
<td>S7</td>
<td>S1</td>
</tr>
<tr>
<td>S14</td>
<td>S8</td>
<td>S2</td>
</tr>
<tr>
<td>S15</td>
<td>S9</td>
<td>S3</td>
</tr>
<tr>
<td>S16</td>
<td>S10</td>
<td>S4</td>
</tr>
<tr>
<td>S17</td>
<td>S11</td>
<td>S5</td>
</tr>
<tr>
<td>S18</td>
<td>S12</td>
<td>S6</td>
</tr>
</tbody>
</table>

To ease problems of routing pressure tubing, the tappings are distributed over six blades:

<table>
<thead>
<tr>
<th>Blade</th>
<th>P(ressure)</th>
<th>S(uction)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>13,9,5</td>
<td>14,10,6</td>
</tr>
<tr>
<td>2</td>
<td>14,10,6</td>
<td>15,11,1</td>
</tr>
<tr>
<td>3</td>
<td>15,11,1</td>
<td>16,12,2</td>
</tr>
<tr>
<td>4</td>
<td>16,12,2</td>
<td>17,7,3</td>
</tr>
<tr>
<td>5</td>
<td>17,17,3</td>
<td>18,8,4</td>
</tr>
<tr>
<td>6</td>
<td>18,8,4</td>
<td>13,9,5</td>
</tr>
</tbody>
</table>

Table 2.1 Positioning of blade surface pressure tappings
<table>
<thead>
<tr>
<th>Test run (OGV type, station)</th>
<th>$x/L$</th>
<th>$\delta_i^*$ (mm)</th>
<th>$\theta_i$ (mm)</th>
<th>$H_i$</th>
<th>$\delta_o^*$ (mm)</th>
<th>$\theta_o$ (mm)</th>
<th>$H_o$</th>
</tr>
</thead>
<tbody>
<tr>
<td>1 ('lo-stag', chevron, W2)</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>&quot; W4</td>
<td>0.5</td>
<td>0.3</td>
<td>1.5</td>
<td>1.2</td>
<td>1.0</td>
<td>1.2</td>
<td></td>
</tr>
<tr>
<td>&quot; W5</td>
<td>0.020</td>
<td>2.6</td>
<td>2.1</td>
<td>1.2</td>
<td>2.4</td>
<td>1.7</td>
<td>1.4</td>
</tr>
<tr>
<td>&quot; W6</td>
<td>0.169</td>
<td>3.9</td>
<td>2.9</td>
<td>1.3</td>
<td>3.3</td>
<td>2.4</td>
<td>1.4</td>
</tr>
<tr>
<td>&quot; W8</td>
<td>0.402</td>
<td>5.2</td>
<td>3.8</td>
<td>1.4</td>
<td>4.2</td>
<td>3.1</td>
<td>1.4</td>
</tr>
<tr>
<td>&quot; W10</td>
<td>0.636</td>
<td>6.4</td>
<td>4.3</td>
<td>1.5</td>
<td>5.7</td>
<td>4.2</td>
<td>1.4</td>
</tr>
<tr>
<td>&quot; W12</td>
<td>1.000</td>
<td>10.1</td>
<td>5.7</td>
<td>1.8</td>
<td>7.3</td>
<td>5.1</td>
<td>1.4</td>
</tr>
<tr>
<td>2 ('lo-stag', straight, W4)</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>&quot; W5</td>
<td>0.020</td>
<td>2.9</td>
<td>2.2</td>
<td>1.3</td>
<td>3.4*</td>
<td>1.7*</td>
<td>2.0*</td>
</tr>
<tr>
<td>&quot; W6</td>
<td>0.169</td>
<td>3.6</td>
<td>2.5</td>
<td>1.4</td>
<td>4.4</td>
<td>2.9</td>
<td>1.5</td>
</tr>
<tr>
<td>&quot; W8</td>
<td>0.402</td>
<td>4.8</td>
<td>3.2</td>
<td>1.5</td>
<td>5.6</td>
<td>3.7</td>
<td>1.5</td>
</tr>
<tr>
<td>&quot; W10</td>
<td>0.636</td>
<td>6.5</td>
<td>4.2</td>
<td>1.5</td>
<td>7.3</td>
<td>4.6</td>
<td>1.6</td>
</tr>
<tr>
<td>&quot; W12</td>
<td>1.000</td>
<td>9.3</td>
<td>5.6</td>
<td>1.7</td>
<td>10.7</td>
<td>6.2</td>
<td>1.7</td>
</tr>
<tr>
<td>3 ('hi-stag', chevron, W2)</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>&quot; W4</td>
<td>3.1</td>
<td>2.2</td>
<td>1.4</td>
<td>5.0</td>
<td>2.5</td>
<td>2.0</td>
<td></td>
</tr>
<tr>
<td>&quot; W5</td>
<td>0.020</td>
<td>4.3</td>
<td>3.4</td>
<td>1.3</td>
<td>1.3</td>
<td>1.1</td>
<td>1.2</td>
</tr>
<tr>
<td>&quot; W6</td>
<td>0.169</td>
<td>4.6</td>
<td>3.7</td>
<td>1.3</td>
<td>1.2</td>
<td>1.0</td>
<td>1.2</td>
</tr>
<tr>
<td>&quot; W8</td>
<td>0.402</td>
<td>5.6</td>
<td>4.2</td>
<td>1.3</td>
<td>1.7</td>
<td>1.4</td>
<td>1.2</td>
</tr>
<tr>
<td>&quot; W10</td>
<td>0.636</td>
<td>7.6</td>
<td>5.3</td>
<td>1.4</td>
<td>2.0</td>
<td>1.6</td>
<td>1.3</td>
</tr>
<tr>
<td>&quot; W12</td>
<td>1.000</td>
<td>9.3</td>
<td>5.9</td>
<td>1.6</td>
<td>2.6</td>
<td>2.1</td>
<td>1.3</td>
</tr>
<tr>
<td>4 ('hi-stag', straight, W2)</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>&quot; W4</td>
<td>3.1</td>
<td>2.1</td>
<td>1.4</td>
<td>4.9</td>
<td>2.2</td>
<td>2.3</td>
<td></td>
</tr>
<tr>
<td>&quot; W5</td>
<td>0.020</td>
<td>4.0</td>
<td>2.9</td>
<td>1.4</td>
<td>1.8</td>
<td>1.4</td>
<td>1.2</td>
</tr>
<tr>
<td>&quot; W6</td>
<td>0.169</td>
<td>5.0</td>
<td>3.5</td>
<td>1.4</td>
<td>2.7</td>
<td>2.2</td>
<td>1.2</td>
</tr>
<tr>
<td>&quot; W8</td>
<td>0.402</td>
<td>5.6</td>
<td>3.9</td>
<td>1.4</td>
<td>3.4</td>
<td>2.6</td>
<td>1.3</td>
</tr>
<tr>
<td>&quot; W10</td>
<td>0.636</td>
<td>8.0</td>
<td>5.1</td>
<td>1.6</td>
<td>4.1</td>
<td>3.1</td>
<td>1.3</td>
</tr>
<tr>
<td>&quot; W12</td>
<td>1.000</td>
<td>11.4</td>
<td>6.8</td>
<td>1.7</td>
<td>4.7</td>
<td>3.5</td>
<td>1.3</td>
</tr>
</tbody>
</table>

* see Section 3.4

Table 4.1 Wall boundary layer parameters
<table>
<thead>
<tr>
<th>Test run (OGV type)</th>
<th>( \Delta \frac{C_p}{5.12} )</th>
<th>( C_p ) 5.12</th>
<th>( C_p' ) 5.12</th>
<th>( e_{5.12} )</th>
<th>( \lambda_{5.12} )</th>
<th>( a_{5} )</th>
<th>( a_{12} )</th>
</tr>
</thead>
<tbody>
<tr>
<td>1 (Lo-stag', chevron)</td>
<td>0.612</td>
<td>0.639</td>
<td>0.594</td>
<td>0.930</td>
<td>0.016</td>
<td>1.089</td>
<td>1.171</td>
</tr>
<tr>
<td>2 (Lo-stag', straight)</td>
<td>0.612</td>
<td>0.639</td>
<td>0.564</td>
<td>0.883</td>
<td>0.047</td>
<td>1.144</td>
<td>1.230</td>
</tr>
<tr>
<td>3 (Hi-stag', chevron)</td>
<td>0.643</td>
<td>0.639</td>
<td>0.573</td>
<td>0.897</td>
<td>0.078</td>
<td>1.125</td>
<td>1.112</td>
</tr>
<tr>
<td>4 (Hi-stag', straight)</td>
<td>0.632</td>
<td>0.639</td>
<td>0.549</td>
<td>0.859</td>
<td>0.090</td>
<td>1.133</td>
<td>1.153</td>
</tr>
<tr>
<td>Ref.95, f.dev. flow</td>
<td>0.584</td>
<td>0.750</td>
<td>0.545</td>
<td>0.727</td>
<td>0.080</td>
<td>1.045</td>
<td>1.740</td>
</tr>
<tr>
<td>Ref.81, chevron</td>
<td>0.740</td>
<td>0.750</td>
<td>0.642</td>
<td>0.856</td>
<td>0.098</td>
<td>1.232</td>
<td>1.281</td>
</tr>
<tr>
<td>Ref.81, straight</td>
<td>0.659</td>
<td>0.750</td>
<td>0.540</td>
<td>0.720</td>
<td>0.119</td>
<td>1.126</td>
<td>1.550</td>
</tr>
</tbody>
</table>

a) Overall diffuser performance

<table>
<thead>
<tr>
<th>Test run (OGV type)</th>
<th>st'n (n)</th>
<th>x/L</th>
<th>( \alpha_n )</th>
<th>( \frac{m_n - m_2}{m_2} )</th>
<th>( C_p ) 5.12</th>
<th>( \lambda_{5-n} )</th>
</tr>
</thead>
<tbody>
<tr>
<td>1 (Lo-stag', chevron)</td>
<td>5</td>
<td>0.020</td>
<td>1.089</td>
<td>0.005</td>
<td>0.000</td>
<td>0.000</td>
</tr>
<tr>
<td></td>
<td>6</td>
<td>0.169</td>
<td>1.101</td>
<td>0.022</td>
<td>0.195</td>
<td>-0.031</td>
</tr>
<tr>
<td></td>
<td>8</td>
<td>0.402</td>
<td>1.102</td>
<td>0.031</td>
<td>0.362</td>
<td>-0.004</td>
</tr>
<tr>
<td></td>
<td>10</td>
<td>0.636</td>
<td>1.113</td>
<td>0.036</td>
<td>0.477</td>
<td>0.010</td>
</tr>
<tr>
<td></td>
<td>12</td>
<td>1.000</td>
<td>1.171</td>
<td>0.037</td>
<td>0.594</td>
<td>0.016</td>
</tr>
<tr>
<td>2 (Lo-stag', straight)</td>
<td>5</td>
<td>0.020</td>
<td>1.144</td>
<td>0.019</td>
<td>0.000</td>
<td>0.000</td>
</tr>
<tr>
<td></td>
<td>6</td>
<td>0.169</td>
<td>1.142</td>
<td>0.006</td>
<td>0.173</td>
<td>-0.001</td>
</tr>
<tr>
<td></td>
<td>8</td>
<td>0.402</td>
<td>1.147</td>
<td>0.005</td>
<td>0.342</td>
<td>0.021</td>
</tr>
<tr>
<td></td>
<td>10</td>
<td>0.636</td>
<td>1.168</td>
<td>0.006</td>
<td>0.455</td>
<td>0.033</td>
</tr>
<tr>
<td></td>
<td>12</td>
<td>1.000</td>
<td>1.230</td>
<td>0.010</td>
<td>0.564</td>
<td>0.047</td>
</tr>
<tr>
<td>3 (Hi-stag', chevron)</td>
<td>5</td>
<td>0.020</td>
<td>1.125</td>
<td>0.001</td>
<td>0.000</td>
<td>0.000</td>
</tr>
<tr>
<td></td>
<td>6</td>
<td>0.169</td>
<td>1.066</td>
<td>-0.006</td>
<td>0.270</td>
<td>-0.056</td>
</tr>
<tr>
<td></td>
<td>8</td>
<td>0.402</td>
<td>1.071</td>
<td>-0.001</td>
<td>0.447</td>
<td>-0.055</td>
</tr>
<tr>
<td></td>
<td>10</td>
<td>0.636</td>
<td>1.090</td>
<td>0.005</td>
<td>0.542</td>
<td>-0.024</td>
</tr>
<tr>
<td></td>
<td>12</td>
<td>1.000</td>
<td>1.112</td>
<td>0.047</td>
<td>0.573</td>
<td>0.078</td>
</tr>
<tr>
<td>4 (Hi-stag', straight)</td>
<td>5</td>
<td>0.020</td>
<td>1.133</td>
<td>-0.007</td>
<td>0.000</td>
<td>0.000</td>
</tr>
<tr>
<td></td>
<td>6</td>
<td>0.169</td>
<td>1.099</td>
<td>-0.057</td>
<td>0.201</td>
<td>-0.011</td>
</tr>
<tr>
<td></td>
<td>8</td>
<td>0.402</td>
<td>1.096</td>
<td>0.009</td>
<td>0.305</td>
<td>0.081</td>
</tr>
<tr>
<td></td>
<td>10</td>
<td>0.636</td>
<td>1.124</td>
<td>-0.010</td>
<td>0.450</td>
<td>0.055</td>
</tr>
<tr>
<td></td>
<td>12</td>
<td>1.000</td>
<td>1.153</td>
<td>0.010</td>
<td>0.549</td>
<td>0.090</td>
</tr>
</tbody>
</table>

b) Performance at stations within diffuser

Table 4.2 Diffuser performance figures
<table>
<thead>
<tr>
<th>Test run (OGV type)</th>
<th>$\alpha_4$</th>
<th>$\alpha_5$</th>
<th>$\frac{m_4 - m_2}{m_2}$</th>
<th>$\frac{m_{12} - m_2}{m_2}$</th>
<th>$C_{p_{4.5}}$</th>
<th>$\lambda_{4-5}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>1 ('lo-stag', chevron)</td>
<td>1.070</td>
<td>1.089</td>
<td>-0.013</td>
<td>0.005</td>
<td>0.151</td>
<td>0.110</td>
</tr>
<tr>
<td>2 ('lo-stag', straight)</td>
<td>1.070</td>
<td>1.144</td>
<td>-0.027</td>
<td>0.019</td>
<td>0.137</td>
<td>0.120</td>
</tr>
<tr>
<td>3 ('hi-stag', chevron)</td>
<td>1.176</td>
<td>1.126</td>
<td>0.040</td>
<td>0.001</td>
<td>0.381</td>
<td>0.111</td>
</tr>
<tr>
<td>4 ('hi-stag', straight)</td>
<td>1.196</td>
<td>1.130</td>
<td>-0.007</td>
<td>-0.007</td>
<td>0.418</td>
<td>0.083</td>
</tr>
<tr>
<td>Ref.81, (chevron)</td>
<td>1.076</td>
<td>1.232</td>
<td>-0.026</td>
<td>-0.037</td>
<td>-0.048</td>
<td>0.190</td>
</tr>
<tr>
<td>Ref.81, (straight)</td>
<td>1.025</td>
<td>1.135</td>
<td>-0.008</td>
<td>0.173</td>
<td></td>
<td></td>
</tr>
<tr>
<td>8 (rainbow - check)</td>
<td>1.171</td>
<td>1.136</td>
<td>-0.026</td>
<td>-0.037</td>
<td>0.338</td>
<td>0.117</td>
</tr>
<tr>
<td>13 (rainbow - hub cl'nce)</td>
<td>1.163</td>
<td>1.078</td>
<td>-0.060</td>
<td>-0.006</td>
<td>0.353</td>
<td>0.127</td>
</tr>
</tbody>
</table>

Table 4.3 OGV performance figures

<table>
<thead>
<tr>
<th>Test run (OGV type)</th>
<th>$\alpha_4$</th>
<th>$\alpha_{12}$</th>
<th>$\frac{m_4 - m_2}{m_2}$</th>
<th>$\frac{m_{12} - m_2}{m_2}$</th>
<th>$C_{p_{4.12}}$</th>
<th>$\lambda_{4-12}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>1 ('lo-stag', chevron)</td>
<td>1.071</td>
<td>1.171</td>
<td>-0.013</td>
<td>0.037</td>
<td>0.590</td>
<td>0.122</td>
</tr>
<tr>
<td>2 ('lo-stag', straight)</td>
<td>1.073</td>
<td>1.230</td>
<td>-0.027</td>
<td>0.010</td>
<td>0.556</td>
<td>0.155</td>
</tr>
<tr>
<td>3 ('hi-stag', chevron)</td>
<td>1.167</td>
<td>1.112</td>
<td>0.040</td>
<td>0.047</td>
<td>0.672</td>
<td>0.150</td>
</tr>
<tr>
<td>4 ('hi-stag', straight)</td>
<td>1.176</td>
<td>1.153</td>
<td>-0.007</td>
<td>0.010</td>
<td>0.692</td>
<td>0.128</td>
</tr>
<tr>
<td>Ref.81, (chevron)</td>
<td>1.076</td>
<td>1.281</td>
<td>-</td>
<td>0.503</td>
<td>0.274</td>
<td></td>
</tr>
<tr>
<td>Ref.81, (straight)</td>
<td>1.025</td>
<td>1.550</td>
<td>-</td>
<td>0.443</td>
<td>0.273</td>
<td></td>
</tr>
</tbody>
</table>

Table 4.4 Combined OGV/Diffuser performance figures
<table>
<thead>
<tr>
<th>Test run (OGV type)</th>
<th>( \frac{\bar{u}'\bar{v}' \Delta^2}{\bar{u}^2} )\text{,}_{i,\text{max}}</th>
<th>( \frac{\bar{u}'\bar{v}' \Delta^2}{\bar{u}^2} )\text{,}_{o,\text{max}}</th>
</tr>
</thead>
<tbody>
<tr>
<td>1 ('lo-stag', chevron)</td>
<td>0.0084</td>
<td>0.0076</td>
</tr>
<tr>
<td>2 ('lo-stag', straight)</td>
<td>0.0068</td>
<td>0.0084</td>
</tr>
<tr>
<td>Ref.95, f.dev. flow, exp.</td>
<td>0.0047</td>
<td>0.0060</td>
</tr>
<tr>
<td>Ref.124, f.dev. flow, pred.</td>
<td>0.0033</td>
<td>N/A</td>
</tr>
</tbody>
</table>

Table 4.5 Diffuser exit Reynolds shear stress figures
FIGURES
a) Fairied diffuser arrangement

A - Pre-diffuser
B - Combustion zone
C - Outer annulus
D - Inner annulus
E - Fuel feed arm

b) Dump diffuser arrangement

Fig. 1.1 Diffuser / Combustor arrangements
Fig. 1.2 Schematic layout of OGV/Diffuser/Combustor
C chord length
d deviation angle
i incidence angle
s space (or pitch)
\(\beta\) air angle
\(\beta'\) blade angle
\(\xi\) stagger angle
\(\chi\) camber angle

Fig. 1.3 Cascade geometry
A data indicating suction surface separation bubble
+ data indicating turbulent separation from suction surface

Fig. 1.4 Blade surface pressure after Masek & Norbury [6],
- effect of separation
Fig. 1.5 Blade surface pressure after Masek & Norbury [6],
effect of incidence variation
Fig. 1.6 Classical secondary flow patterns
Fig. 1.7 Corner flow directions for different flow regimes

a) Laminar corner flow

b) Turbulent corner flow
Fig. 1.8 Rotor blade passage vortices after Inoue & Kuroumaru [36]
a) Blade sweep: flow direction is not perpendicular to spanwise direction
e.g.-

i)  

ii)  

b) Blade dihedral: blade surface not perpendicular to end-wall
e.g.-

Fig. 1.9 Description of blade sweep and dihedral
a) straight blade (no dihedral)

b) leant blade (constant dihedral)

c) chevron blade (double dihedral)

d) curved blade (continuously varying dihedral)

Fig. 1.10 Different forms of blade dihedral
Inlet blockage = 0.02

Fig.1.11 Annular diffuser performance after Sovran & Klomp [98]
Fig. 1.12 Diffuser data after Howard et. al. [97]

a) Lines of first stall

\[
\frac{L}{\Delta r_{\text{inlet}}}
\]

AR - 1

b) Pressure recovery for equiangular annular diffusers (fully developed inflow)

\[
\frac{W}{\Delta r_{\text{inlet}}}
\]
dimensions given are approximate

Fig. 2.1 Test rig diagram
a) Axial positioning

b) Circumferential positioning

Fig. 2.2 Traverse station positioning
For further details of all traverse positions, see Fig.2.2

Fig.2.3 Probe location at Station 4 for 'low stagger' tests
For further details of all traverse positions, see Fig.2.2

Fig.2.4 Probe location at Station 4 for 'high stagger' tests
a) Pitot probe
  i/d - 0.66mm

b) Five-hole probe
  i/d - 0.25mm

Fig. 2.5 Dimensions of pressure probes
Tangents to camber line, at 30% chord

Traverse / results planes (spaced 10% chord intervals)

a velocity vector in plane of paper
b component of 'a' shown in Fig. 4.2.8
c component of 'a' shown in Fig. 4.2.12

Fig. 2.6 Interblade traversing geometry and definitions
Fig. 3.1 Circumferential distribution of axial velocity at rotor inlet
- Run 1: 'low stagger' chevron
Fig.3.2a) Circumferential distribution of axial velocity at OGV exit
- Run 1: 'low stagger' chevron
Fig. 3.2b) Circumferential distribution of axial velocity at OGV exit
- Run 2: 'low stagger' straight
Fig. 3.3a) Circumferential distribution of axial velocity at diffuser exit
- Run 1: 'low stagger' chevron
Fig. 3.3b) Circumferential distribution of axial velocity at diffuser exit
- Run 2: 'low stagger' straight
Fig. 3.4a) Circumferential distribution of mass flow error
- Run 1: 'low stagger' chevron
Fig. 3.4b) Circumferential distribution of mass flow error
- Run 2: 'low stagger' straight
Fig. 3.5 Circumferential distribution of axial velocity at rotor inlet
- Run 3: 'high stagger' chevron
Fig. 3.6a) Circumferential distribution of axial velocity at OGV exit
- Run 3: 'high stagger' chevron
Fig. 3.6b) Circumferential distribution of axial velocity at OGV exit
- Run 4: 'high stagger' straight
Fig. 3.7a) Circumferential distribution of axial velocity at diffuser exit
- Run 3: 'high stagger' chevron
Fig. 3.7b) Circumferential distribution of axial velocity at diffuser exit
- Run 4: 'high stagger' straight
Fig. 3.8a) Circumferential distribution of mass flow error
- Run 3: 'high stagger' chevron
Fig. 3.8b) Circumferential distribution of mass flow error
- Run 4: 'high stagger' straight
Fig. 4.1.1 Axial velocity profile at rotor inlet
- Run 1: 'low stagger' chevron
Fig. 4.1.2 Rotor incidence

- Run 1: 'low stagger' chevron
Fig. 4.1.3 Axial velocity profile at rotor exit
- Run 1,2: 'low stagger' chevron & straight
Fig. 4.1.4 Rotor deviation
- Run 1, 2: 'low stagger' chevron & straight
Fig. 4.1.5 OGV incidence
- Run 1,2: 'low stagger' chevron & straight
Fig. 4.1.6 Axial velocity profile at OGV exit
- Run 1, 2: 'low stagger' chevron & straight
Fig. 4.1.7 Axial velocity profile at OGV exit after Harasgama [81]
Fig. 4.1.8a) Axial velocity contours at OGV exit
- Run 1: 'low stagger' chevron

Contours are of constant $\frac{u}{\bar{u}}$
Fig. 4.1.8b) Axial velocity contours at OGV exit
- Run 2: 'low stagger' straight

Contours are of constant $\frac{u}{\bar{u}}$
Fig. 4.1.9a) Velocity vectors at OGV exit
- Run 1: 'low stagger' chevron
Fig. 4.1.9b) Velocity vectors at OGV exit
- Run 2: 'low stagger' straight
Fig. 4.1.10a) Velocity vectors at OGV exit - radial (diffuser curvature) component removed
- Run 1: 'low stagger' chevron
Fig. 4.1.10b) Velocity vectors at OGV exit - radial (diffuser curvature) component removed
- Run 2: 'low stagger' straight
Pitch-averaged yaw angle (deg.)
- relative to axial

$\bar{\gamma}$

Fig. 4.1.11 Yaw angles at OGV exit
- Run 1,2: 'low stagger' chevron & straight
Contours are of constant \( \frac{P_{w5} - P_{w4}}{(P_t - \bar{p})_{w4}} \)

Fig.4.1.12a) OGV loss contours

- Run 1: 'low stagger' chevron
Contours are of constant 
\[ \frac{(P_{W5} - P_{W4})}{(P - \bar{P})_{W4}} \]

Fig. 4.1.12b) OGV loss contours
- Run 2: 'low stagger' straight
Fig. 4.2.1a) Axial velocity profile at rotor inlet
- Run 3, 4: 'high stagger' chevron & straight
Fig. 4.2.1b) Axial velocity profile at rotor inlet for Run 3,4
c.f. embedded stage velocity profiles due to Howell [87], Smith [88]
Fig. 4.2.2 Rotor incidence
- Run 3, 4: 'high stagger' chevron & straight
see Fig. 2.4 for probe axial location

Fig. 4.2.3 Axial velocity profile at rotor exit
- Run 3,4: 'high stagger' chevron & straight
Fig. 4.2.4 Absolute velocity profile at rotor exit
- Run 3, 4: 'high stagger' chevron & straight
Fig. 4.2.5 Rotor deviation
- Run 3, 4: 'high stagger' chevron & straight
Run 3: 'high stagger' chevron, 5-hole probe
Run 4: 'high stagger' straight, 5-hole probe
Run 4: 'high stagger' straight, 3-hole probe
estimate at OGV l/e

see Fig. 2.4 for probe axial location

Fig. 4.2.6 OGV incidence
- Run 3,4: 'high stagger' chevron & straight
Fig. 4.27a) Axial velocity profiles through OGV passage

- Run 4: 'high stagger' straight
**Fig. 4.27b** Axial velocity profiles through OGV passage

- Run 4: 'high stagger' straight

- W4 traverse (37% $C_x$ upstream)
- 0% chord
- 100% chord
- W5 traverse (13% $C_x$ downstream)
Data outside thick lines is extrapolated

Fig. 4.2.8 Velocity relief maps through OGV passage
- Run 4: 'high stagger' straight
Data outside thick lines is extrapolated

Fig. 4.2.9 Axial velocity relief maps through OGV passage
- Run 4: 'high stagger' straight
Data outside thick lines is extrapolated

Fig. 4.2.10 Stagnation pressure relief maps through OGV passage
- Run 4: 'high stagger' straight
Data outside thick lines is extrapolated

Fig. 4.2.11 Static pressure relief maps through OGV passage
- Run 4: 'high stagger' straight
Fig. 4.2.12 Velocity vectors through OGV passage
- Run 4: 'high stagger' straight
Fig. 4.2.13 Diagram showing OGV passage vortices
Pitch-averaged yaw angle (deg.)
- relative to tangent with blade camber

Fig. 4.2.14 Yaw angles through OGV passage
- Run 4: 'high stagger' straight
Fig. 4.2.15a) OGV surface pressure distribution
- Run 3: 'high stagger' chevron
Fig.4.2.15b) OGV surface pressure distribution
- Run 4: 'high stagger' straight
Points connected by lines are from blade surface pressure tappings
Discrete points are linearly extrapolated from inter-blade traverse data

\[
\frac{p - \bar{p}_{W4}}{(\bar{p}_t - \bar{p})_{W4}}
\]

Fig. 4.2.15c) OGV surface pressure distribution
- Run 4: 'high stagger' straight
a) straight blade (no dihedral)

b) leant blade (constant dihedral)

c) chevron blade (double dihedral)

Fig. 4.2.16 Blade passage pressure distribution for dihedral blade
Run 3: 'high stagger' chevron
Run 4: 'high stagger' straight

Fig.4.2.17 Axial velocity profile at OGV exit
- Run 3,4: 'high stagger' chevron & straight
Contours are of constant $\frac{u}{u}$

Fig. 4.2.18a) Axial velocity contours at OGV exit
- Run 3: 'high stagger' chevron
Contours are of constant $\frac{u}{u}$

Fig.4.2.18b) Axial velocity contours at OGV exit
- Run 4: 'high stagger' straight
Fig. 4.2.19 Tangential blockage at OGV exit
- Run 3, 4: 'high stagger' chevron & straight
Fig. 4.2.20a) Velocity vectors at OGV exit
- Run 3: 'high stagger' chevron
Fig. 4.2.20b) Velocity vectors at OGV exit
- Run 4: 'high stagger' straight
Fig. 4.2.21a) Velocity vectors at OGV exit - radial (diffuser curvature) component removed
- Run 3: 'high stagger' chevron
Fig. 4.2.21b) Velocity vectors at OGV exit - radial (diffuser curvature) component removed
- Run 4: 'high stagger' straight
Fig. 4.2.22 Velocity vectors at 100% OGV chord - from inter-blade traverse - Run 4: 'high stagger' straight
Pitch-averaged yaw angle (deg.) - relative to axial

Fig. 4.2.23 Yaw angles at OGV exit
- Run 3,4: 'high stagger' chevron & straight
Contours are of constant \( \frac{(P_{w5} - P_{w4})}{(\bar{P}_t - \bar{p})_{w4}} \)

Fig.4.2.24a) OGV loss contours
- Run 3: 'high stagger' chevron
Contours are of constant \( \frac{(P_{W5} - P_{W4})}{(P_{T} - \bar{p})_{W4}} \)

Fig.4.2.24b) OGV loss contours

- Run 4: 'high stagger' straight
Fig. 4.3.1 Axial velocity profile at OGV exit

- Run 3, 8: 'high stagger' chevron & 'rainbow' control test
Contours are of constant $\frac{u}{u}$

Fig.4.3.2 Axial velocity contours at OGV exit
- Run 8: 'rainbow' control test
Fig. 4.3.3a) Axial velocity contours at OGV exit after Harasgama\cite{81}
- chevron blade.
Contours are of constant $\frac{u}{\bar{u}}$

Fig. 4.3.3b) Axial velocity contours at OGV exit after Harasgama$^{[81]}$ - straight blade
Contours are of constant $\frac{u}{u}$

Fig. 4.3.4 Axial velocity contours at OGV exit
- Run 6: low Reynolds number test
Fig. 4.3.5 Axial velocity profile at OGV exit
- Run 4, 13: 'high stagger' straight & hub clearance test
Contours are of constant $\frac{u}{u}$

Fig. 4.3.6 Axial velocity contours at OGV exit
- Run 13: hub clearance test
Fig. 4.3.7 Tangential blockage at OGV exit
- Run 4, 13: 'high stagger' straight & hub clearance test
Fig. 4.4.1 Axial velocity profile at diffuser inlet
- Run 1-4: 'low' & 'high stagger', chevron & straight
Fig. 4.4.2 Yaw angle at diffuser inlet
- Run 1-4: 'low' & 'high stagger', chevron & straight
Fig. 4.4.3 Axial velocity profile at diffuser exit

- Run 1-4: 'low' & 'high stagger', chevron & straight
Fig. 4.4.4a) Axial velocity profile down diffuser
- Run 1: 'low stagger' chevron
Fig. 4.4.4b) Axial velocity profile down diffuser
- Run 2: 'low stagger' straight
Fig. 4.4.4c) Axial velocity profile down diffuser
- Run 3: 'high stagger' chevron
Fig. 4.4.4d) Axial velocity profile down diffuser
- Run 4: 'high stagger' straight
Fig. 4.4.5a) Diffuser inner wall boundary layer displacement thickness
- Run 1-4: 'low' & 'high stagger', chevron & straight
Run 1: 'low stagger' chevron
Run 2: 'low stagger' straight
Run 3: 'high stagger' chevron
Run 4: 'high stagger' straight

Fig. 4.4.5b) Diffuser inner wall boundary layer momentum thickness
- Run 1-4: 'low' & 'high stagger', chevron & straight
Fig. 4.4.5c) Diffuser inner wall boundary layer shape parameter
- Run 1-4: 'low' & 'high stagger', chevron & straight
Fig. 4.4.6a) Diffuser outer wall boundary layer displacement thickness
- Run 1-4: 'low' & 'high stagger', chevron & straight
Run 1: 'low stagger' chevron
Run 2: 'low stagger' straight
Run 3: 'high stagger' chevron
Run 4: 'high stagger' straight

Fig.4.4.6b) Diffuser outer wall boundary layer momentum thickness
Run 1-4: 'low' & 'high stagger', chevron & straight
Fig. 4.4.6c) Diffuser outer wall boundary layer shape parameter

- Run 1-4: 'low' & 'high stagger', chevron & straight
Contours are of constant $\frac{u}{u}$

Fig.4.4.7a) Diffuser axial velocity contours at $x/L=0.020$
- Run 1: 'low stagger' chevron
Contours are of constant $\frac{u}{u}$

Fig. 4.4.7b) Diffuser axial velocity contours at $x/L=0.169$
- Run 1: 'low stagger' chevron
Contours are of constant $\frac{u}{\bar{u}}$

Fig.4.4.7c) Diffuser axial velocity contours at x/L=0.402
- Run 1: 'low stagger' chevron
Contours are of constant $\frac{u}{u}$

Fig. 4.4.7d) Diffuser axial velocity contours at x/L=0.636
- Run 1: 'low stagger' chevron
Contours are of constant \( \frac{u}{u} \)

Fig. 4.1.7e) Diffuser axial velocity contours at \( x/L = 1.000 \)
- Run 1: 'low stagger' chevron
Contours are of constant \( \frac{u}{u} \)

Fig. 4.4.8a) Diffuser axial velocity contours at \( x/L=0.020 \)
- Run 2: 'low stagger' straight
Contours are of constant $\frac{u}{\bar{u}}$

Fig. 4.4.8b) Diffuser axial velocity contours at $x/L=0.169$
- Run 2: 'low stagger' straight
Fig. 4.4.8c) Diffuser axial velocity contours at $x/L = 0.402$
- Run 2: 'low stagger' straight

Contours are of constant $\frac{u}{u}$
Contours are of constant $\frac{u}{\bar{u}}$

Fig. 4.4.8d) Diffuser axial velocity contours at $x/L = 0.636$
- Run 2: 'low stagger' straight
Contours are of constant $\frac{u}{u}$

Fig. 4.4.8e) Diffuser axial velocity contours at $x/L=1.000$
- Run 2: 'low stagger' straight
Contours are of constant \( \frac{u}{u} \)

Fig. 4.4.9a) Diffuser axial velocity contours at \( x/L = 0.020 \)
- Run 3: 'high stagger' chevron
Contours are of constant $\frac{u}{u}$

Fig. 4.4.9b) Diffuser axial velocity contours at x/L=0.169
- Run 3: 'high stagger' chevron
Contours are of constant $\frac{u}{\bar{u}}$

Fig.4.4.9c) Diffuser axial velocity contours at $x/L=0.402$

- Run 3: 'high stagger' chevron
Contours are of constant $\frac{u}{u}$

Fig. 4.4.9d) Diffuser axial velocity contours at $x/L=0.636$
- Run 3: 'high stagger' chevron
Fig. 4.4.9e) Diffuser axial velocity contours at x/L=1.000
- Run 3: 'high stagger' chevron

Contours are of constant $\frac{u}{\bar{u}}$
Contours are of constant $\frac{u}{u}$

Fig. 4.4.10a) Diffuser axial velocity contours at $x/L=0.020$
- Run 4: 'high stagger' straight
Contours are of constant \( \frac{u}{u} \)

Fig. 4.4.10b) Diffuser axial velocity contours at \( x/L = 0.169 \)
- Run 4: 'high stagger' straight
Contours are of constant $\frac{u}{u}$

Fig. 4.4.10c) Diffuser axial velocity contours at $x/L=0.402$
- Run 4: 'high stagger' straight
Contours are of constant $\frac{u}{u}$

Fig.4.4.10d) Diffuser axial velocity contours at $x/L=0.636$
- Run 4: 'high stagger' straight
Contours are of constant $\frac{u}{u}$

Fig.4.4.10e) Diffuser axial velocity contours at $x/L=1.000$
- Run 4: 'high stagger' straight
Fig. 4.4.11a) Radial variation of OGV wake prominence
- Run 1: 'low stagger' chevron
Fig. 4.4.11b) Radial variation of OGV wake prominence
- Run 2: 'low stagger' straight
Fig. 4.4.11c) Radial variation of OGV wake prominence
- Run 3: 'high stagger' chevron
Fig. 4.4.11d) Radial variation of OGV wake prominence
- Run 4: 'high stagger' straight
Fig. 4.4.12a) Axial decay of OGV wake
- Run 1: 'low stagger' chevron
Fig. 4.4.12b) Axial decay of OGV wake
- Run 2: 'low stagger' straight
Fig. 4.4.12c) Axial decay of OGV wake
- Run 3: 'high stagger' chevron
Fig. 4.4.12d) Axial decay of OGV wake
- Run 4: 'high stagger' straight
Fig. 4.4.13a) Axial decay of OGV wake after Harasgama [81] - chevron blade
Fig. 4.4.13b) Axial decay of OGV wake after Harasgama [81]
- straight blade
Fig. 4.4.14a) Diffuser pressure recovery from traverse results
- Run 1-4: 'low' & 'high stagger', chevron & straight
Fig. 4.4.14b) Diffuser pressure recovery from outer wall tappings
- Run 1-4: 'low' & 'high stagger', chevron & straight
Fig. 4.4.15 Measured diffuser exit Reynolds shear stress after Stevens [95]
Fig.4.4.16a) Calculated Reynolds shear stress down diffuser
- Run 1: 'low stagger' chevron
Fig. 4.4.16b) Calculated Reynolds shear stress down diffuser
- Run 2: 'low stagger' straight
Fig. 4.4.17a) Terms of streamwise momentum equation at x/L=0.095
- Run 1: 'low stagger' chevron
Fig. 4.4.17b) Terms of streamwise momentum equation at $x/L=0.286$
- Run 1: 'low stagger' chevron
Fig. 4.4.17c) Terms of streamwise momentum equation at x/L=0.519
- Run 1: 'low stagger' chevron
Fig. 4.4.17d) Terms of streamwise momentum equation at x/L=0.818
- Run 1: 'low stagger' chevron
Fig. 4.4.18a) Terms of streamwise momentum equation at $x/L=0.095$
- Run 2: 'low stagger' straight
Fig. 4.4.18b) Terms of streamwise momentum equation at x/L=0.286
- Run 2: 'low stagger' straight
Fig. 4.4.18c) Terms of streamwise momentum equation at x/L=0.519
- Run 2: 'low stagger' straight
Fig. 4.4.18d) Terms of streamwise momentum equation at x/L=0.818
- Run 2: 'low stagger' straight
APPENDICES

A1  Blade Design
A2  Inlet Air Filters
A3  Pressure Transducers
A4  Computer Hardware
A5  Five-Hole Probes
A6  Computer Programs
A7  Photographs
APPENDIX 1 BLADE DESIGN

A1.1 'Low Stagger' Blades
   A1.1.1 Mean Diameter Design Data: Rotor & OGV
   A1.1.2 Estimation of OGV Loss Coefficient

A1.2 'High Stagger' Blades
   A1.2.1 Mean Diameter Design Data: Rotor
   A1.2.2 Mean Diameter Design Data: OGV
   A1.2.3 Estimation of OGV Loss Coefficient
A1.1 'Low Stagger' Blades

A1.1.1 Mean Diameter Design Data: Rotor & OGV

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\beta_1$</td>
<td>air inlet angle</td>
<td>30°</td>
</tr>
<tr>
<td>$\beta_2$</td>
<td>air outlet angle</td>
<td>0°</td>
</tr>
<tr>
<td>$d$</td>
<td>deviation angle</td>
<td>6°</td>
</tr>
<tr>
<td>$\beta_1'$</td>
<td>blade inlet angle</td>
<td>30°</td>
</tr>
<tr>
<td>$\beta_2'$</td>
<td>blade outlet angle</td>
<td>-6°</td>
</tr>
<tr>
<td>$\xi$</td>
<td>stagger angle</td>
<td>12°</td>
</tr>
<tr>
<td>$\chi$</td>
<td>camber angle</td>
<td>36°</td>
</tr>
<tr>
<td>$\epsilon$</td>
<td>deflection angle</td>
<td>30°</td>
</tr>
<tr>
<td>profile</td>
<td>thickness/chord ratio</td>
<td>0.1</td>
</tr>
<tr>
<td>camber line</td>
<td>blade angle variation</td>
<td>circular arc</td>
</tr>
<tr>
<td>$C$</td>
<td>true chord</td>
<td>41.40 mm</td>
</tr>
<tr>
<td>$C_{ax}$</td>
<td>axial chord</td>
<td>40.50 mm</td>
</tr>
<tr>
<td>$D_{eq}$</td>
<td>equivalent diffusion factor (Eqn 1.2.1) - rotor</td>
<td>1.446</td>
</tr>
<tr>
<td></td>
<td>- OGV</td>
<td>1.440</td>
</tr>
<tr>
<td>$s$</td>
<td>space (pitch)</td>
<td>20.73 mm</td>
</tr>
<tr>
<td>$s/C$</td>
<td>space/chord ratio</td>
<td>19.92 mm</td>
</tr>
<tr>
<td>no. of blades</td>
<td>- rotor</td>
<td>0.501</td>
</tr>
<tr>
<td></td>
<td>- OGV</td>
<td>0.481</td>
</tr>
<tr>
<td>$U$</td>
<td>rotor speed</td>
<td>27.62 ms$^{-1}$ (650 rpm)</td>
</tr>
<tr>
<td>$u$</td>
<td>axial velocity</td>
<td>47.84 ms$^{-1}$</td>
</tr>
<tr>
<td>$u/U$</td>
<td>flow coefficient</td>
<td>1.732</td>
</tr>
<tr>
<td>$\Lambda$</td>
<td>reaction</td>
<td>50%</td>
</tr>
<tr>
<td>$m$</td>
<td>mass flow</td>
<td>7.75 kg sec$^{-1}$</td>
</tr>
<tr>
<td>$Re$</td>
<td>blade chord Reynolds number</td>
<td>1.57 x 10$^5$</td>
</tr>
<tr>
<td>$\Delta T$</td>
<td>stage temperature rise</td>
<td>0.76°C</td>
</tr>
<tr>
<td>$C_p,\Delta T/U^2$</td>
<td>blade loading</td>
<td>1.0</td>
</tr>
<tr>
<td>$h$</td>
<td>blade height</td>
<td>51.82 mm</td>
</tr>
<tr>
<td>$h/C$</td>
<td>aspect ratio</td>
<td>1.25</td>
</tr>
<tr>
<td>$r_{t}/r_o$</td>
<td>hub/tip radius ratio</td>
<td>0.88</td>
</tr>
<tr>
<td>tip clearance</td>
<td></td>
<td>0.76 mm (1.47% of $h$)</td>
</tr>
</tbody>
</table>
Δx_{rs} \quad \text{axial gap between rotor t/e and OGV l/e} \quad 31.09 \text{ mm}

\text{mean diameter} \quad 811.78 \text{ mm}

\text{estimated stalling incidence (Wilson [2])} \quad +20°

A1.1.2 Estimation of OGV Loss Coefficient

The equations and data used in this calculation are taken from Cohen et. al.[113], and Horlock[3].
Nominal incidence is taken as 0°.

\[ \beta_m = \tan^{-1} \left( 0.5 \times (\tan \beta_1 + \tan \beta_2) \right) \]

\[ = \tan^{-1} \left( 0.5 \times (\tan 30° + \tan 0°) \right) \]

\[ = 16.1° \]

Now \[ C_D = C_D(\text{profile}) + C_D(\text{annulus}) + C_D(\text{secondary}) \]

\[ C_D(\text{annulus}) = 0.020 \times \left( \frac{s}{h} \right) \]

\[ = 0.020 \times \frac{19.92}{51.82} \]

\[ = 0.00769 \]

\[ C_L = 2.(\frac{s}{c})(\tan \beta_1 - \tan \beta_2).\cos \beta_m \]

\[ = 2 \times 0.481(\tan 30° - \tan 0°).\cos 16.1° \]

\[ = 0.534 \]

\[ C_D(\text{secondary}) = 0.018 \left( C_L \right)^2 = 0.018 \times (0.534)^2 \]

\[ = 0.00513 \]

\[ C_D(\text{profile}) = 0.02 \]
\[ \therefore C_D = 0.00769 + 0.02 + 0.00513 \]
\[ = 0.0328 \]

\[ \lambda = \frac{C_D}{(\frac{h}{C}) \cos^3 \beta_m} \cos^2 \beta_1 \]
\[ = \frac{0.0328 \cos^2 30^\circ}{0.481 \cos^3 16.1^\circ} \]
\[ = 0.058 \]

A1.2 'High Stagger' Blades

A1.2.1 Mean Diameter Design Data: Rotor

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>( \beta_1 )</td>
<td>air inlet angle</td>
<td>67°</td>
</tr>
<tr>
<td>( \beta_2 )</td>
<td>air outlet angle</td>
<td>56.1°</td>
</tr>
<tr>
<td>( d )</td>
<td>deviation angle</td>
<td>5.1°</td>
</tr>
<tr>
<td>( \beta_1' )</td>
<td>blade inlet angle</td>
<td>67°</td>
</tr>
<tr>
<td>( \beta_2' )</td>
<td>blade outlet angle</td>
<td>51.0°</td>
</tr>
<tr>
<td>( \xi )</td>
<td>stagger angle</td>
<td>59.0°</td>
</tr>
<tr>
<td>( \chi )</td>
<td>camber angle</td>
<td>16°</td>
</tr>
<tr>
<td>( \epsilon )</td>
<td>deflection angle</td>
<td>10.9°</td>
</tr>
<tr>
<td>C</td>
<td>true chord</td>
<td>78.62 mm</td>
</tr>
<tr>
<td>C_{ax}</td>
<td>axial chord</td>
<td>40.49 mm</td>
</tr>
<tr>
<td>D_{eq}</td>
<td>equivalent diffusion factor (Eqn 1.2.1)</td>
<td>1.712</td>
</tr>
<tr>
<td>s</td>
<td>space (pitch)</td>
<td>77.28 mm</td>
</tr>
<tr>
<td>s/C</td>
<td>space/chord ratio</td>
<td>0.983</td>
</tr>
<tr>
<td>no. of blades</td>
<td></td>
<td>33</td>
</tr>
<tr>
<td>U</td>
<td>rotor speed</td>
<td>85.0 ms(^{-1}) (2000 rpm)</td>
</tr>
<tr>
<td>u</td>
<td>axial velocity</td>
<td>36.09 ms(^{-1})</td>
</tr>
<tr>
<td>u/U</td>
<td>flow coefficient</td>
<td>0.425</td>
</tr>
<tr>
<td>( \Lambda )</td>
<td>reaction</td>
<td>81.5%</td>
</tr>
<tr>
<td>Symbol</td>
<td>Description</td>
<td>Value</td>
</tr>
<tr>
<td>--------</td>
<td>--------------------------------------------------</td>
<td>------------------------</td>
</tr>
<tr>
<td>( m )</td>
<td>mass flow</td>
<td>5.85 kg sec(^{-1})</td>
</tr>
<tr>
<td>( Re )</td>
<td>blade chord Reynolds number</td>
<td>4.97 \times 10^5</td>
</tr>
<tr>
<td>( \Delta T )</td>
<td>stage temperature rise</td>
<td>2.65°C</td>
</tr>
<tr>
<td>( C_p \Delta T/U^2 )</td>
<td>blade loading</td>
<td>0.369</td>
</tr>
<tr>
<td>( h )</td>
<td>blade (passage) height</td>
<td>51.82 mm</td>
</tr>
<tr>
<td>( h/C )</td>
<td>aspect ratio</td>
<td>0.659</td>
</tr>
<tr>
<td>( r_i/r_o )</td>
<td>hub/tip radius ratio</td>
<td>0.88</td>
</tr>
<tr>
<td></td>
<td>tip clearance</td>
<td>0.54 mm</td>
</tr>
<tr>
<td>( \Delta x_{rs} )</td>
<td>axial gap between rotor t/e and OGV t/e</td>
<td>14.0 mm</td>
</tr>
<tr>
<td></td>
<td>mean diameter</td>
<td>811.78 mm</td>
</tr>
<tr>
<td></td>
<td>estimated stalling incidence (Wilson [2])</td>
<td>+6(^\circ)</td>
</tr>
</tbody>
</table>

**A1.2.2 Mean Diameter Design Data: OGV**

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>( \beta_1 )</td>
<td>air inlet angle</td>
<td>41.0(^\circ)</td>
</tr>
<tr>
<td>( \beta_2 )</td>
<td>air outlet angle</td>
<td>0.0(^\circ)</td>
</tr>
<tr>
<td>( d )</td>
<td>deviation angle</td>
<td>8.0(^\circ)</td>
</tr>
<tr>
<td>( \beta_1' )</td>
<td>blade inlet angle</td>
<td>41(^\circ)</td>
</tr>
<tr>
<td>( \beta_2' )</td>
<td>blade outlet angle</td>
<td>-8.0(^\circ)</td>
</tr>
<tr>
<td>( \xi )</td>
<td>stagger angle</td>
<td>16.5(^\circ)</td>
</tr>
<tr>
<td>( \chi )</td>
<td>camber angle</td>
<td>49.0(^\circ)</td>
</tr>
<tr>
<td>( \epsilon )</td>
<td>deflection angle</td>
<td>41.0(^\circ)</td>
</tr>
<tr>
<td>( C )</td>
<td>true chord</td>
<td>C5</td>
</tr>
<tr>
<td>( C_{ax} )</td>
<td>axial chord</td>
<td>53.87 mm</td>
</tr>
<tr>
<td>( D_{eq} )</td>
<td>equivalent diffusion factor (Eqn 1.2.1)</td>
<td>1.690</td>
</tr>
<tr>
<td>( s )</td>
<td>space (pitch)</td>
<td>27.72 mm</td>
</tr>
<tr>
<td>( s/C )</td>
<td>space/chord ratio</td>
<td>0.515</td>
</tr>
<tr>
<td></td>
<td>no. of blades</td>
<td>92</td>
</tr>
<tr>
<td>( u )</td>
<td>axial velocity</td>
<td>36.09 ms(^{-1})</td>
</tr>
<tr>
<td>( A )</td>
<td>reaction</td>
<td>81.5%</td>
</tr>
<tr>
<td>( m )</td>
<td>mass flow</td>
<td>5.85 kg sec(^{-1})</td>
</tr>
<tr>
<td>( Re )</td>
<td>blade chord Reynolds number</td>
<td>1.76 \times 10^5</td>
</tr>
<tr>
<td>( h )</td>
<td>blade (passage) height</td>
<td>51.82 mm</td>
</tr>
<tr>
<td>Parameter</td>
<td>Value</td>
<td></td>
</tr>
<tr>
<td>-----------------------------------------------</td>
<td>---------------</td>
<td></td>
</tr>
<tr>
<td>$h/C$ aspect ratio</td>
<td>0.962</td>
<td></td>
</tr>
<tr>
<td>$r_i/r_o$ hub/tip radius ratio</td>
<td>0.88</td>
<td></td>
</tr>
<tr>
<td>$\Delta x_{rs}$ axial gap between rotor t/e and OGV l/e</td>
<td>14.0 mm</td>
<td></td>
</tr>
<tr>
<td>mean diameter</td>
<td>811.78 mm</td>
<td></td>
</tr>
<tr>
<td>estimated stalling incidence (Wilson [2])</td>
<td>+14°</td>
<td></td>
</tr>
</tbody>
</table>

A1.2.3 Estimation of OGV Loss Coefficient

The equations and data used in this calculation are taken from Cohen et. al.[113], and Horlock[3].
Nominal incidence is taken as 0°.

$$\beta_m = \tan^{-1}(0.5 \times (\tan\beta_1 + \tan\beta_2))$$
$$= \tan^{-1}(0.5 \times (\tan41° + \tan0°))$$
$$= 23.5°$$

Now

$$C_D = C_{D(profile)} + C_{D(annulus)} + C_{D(secondary)}$$

$$C_{D(annulus)} = 0.020 \times \left(\frac{s}{h}\right)$$
$$= 0.020 \times \frac{27.72}{51.82}$$
$$= 0.0107$$

$$C_L = 2 \times (\frac{s}{c}) (\tan\beta_1 - \tan\beta_2) \cos \beta_m$$
$$= 2 \times 0.515 (\tan41° - \tan0°) \cos 23.5°$$
$$= 0.820$$

$$C_{D(secondary)} = 0.018 (C_L)^2 = 0.018 (0.820)^2$$
$$= 0.012$$

$$C_{D(profile)} = 0.02$$
\[ \therefore C_D = 0.012 + 0.02 + 0.0107 \]
\[ = 0.0427 \]

\[ \lambda = \frac{C_D \cdot \cos^2 \beta_1}{\left( \frac{s}{C} \right) \cos^3 \beta_m} \]
\[ = \frac{0.0427 \cdot \cos^2 41^\circ}{0.515 \cdot \cos^3 23.5^\circ} \]
\[ = 0.061 \]
APPENDIX 2 INLET AIR FILTERS

A2.1 Manufacturers Specification

A2.2 Usage
A2.1 Manufacturers Specification

Manufacturer: AAF (American Air Filters) Ltd.
Type: DRI-Pak extended surface filters - 41.09.937
Flow rate per unit: 3400 - 4250 - 5100 m³/hr⁻¹
Working pressure differential: 100 - 152 - 200 Nm⁻²

N.B. Filter should be discarded when pressure differential reaches 250 Nm⁻² due to dirt accumulation.

A2.2 Usage

The arrangement of the filter bags within the inlet plenum chamber is indicated by Fig. A2.1. Since the filters have a fairly narrow operating range of mass flow, when the rig is being operated at mass flow rates lower than the minimum allowable with all the bags in operation, some of the bags must be blanked off. The number of open filter bags used for the 'high stagger' tests was calculated as follows.

Converting specification flow rate to a mass flow rate gives a mass flow range per bag of 1.16 - 1.45 - 1.74 kg sec⁻¹

∴ Total mass flow capability = 13.92 - 17.40 - 20.88 kg sec⁻¹

Now,

2-d design mass flow rate = 5.85 kg sec⁻¹

The actual rig mass flow rate will be lower than this due to shear in the wall boundary layers.

∴ Estimated actual mass flow rate

= 2-d design mass flow rate x estimated u/u
= 5.85 x 0.92
= 5.38 kg sec⁻¹

∴ No. of filter bags required = (5.38 / 1.16) - (5.38 / 1.45) - (5.38 / 1.74)
= 4.6 - 3.7 - 3.1 bags open

i.e. either 3.5, 4 or 4.5 filter bags open

In order to provide the maximum run-time for one filter unit arrangement before clogging necessitates replacement, it was decided to run with 4.5 bags open. The selection of which of the bags to blank-off is also important in order to provide a clean air flow path into the rig entry flare. Therefore units 2, 3, 5 and 8 (see Fig. A2.1) were kept open; thus leaving units 1, 4, 6, 7, 9, 10, 11 to be blanked-off. This arrangement was used for all the tests reported in Sections 4.2, 4.3. For the 'low stagger' test (Section 4.1) no filters were installed.
Fig. A2.1 Filter arrangement in inlet plenum chamber
APPENDIX 3 PRESSURE TRANSUCERS

A3.1 Manufacturers Specification

A3.2 Calibration
A3.1 Manufacturers Specification

<table>
<thead>
<tr>
<th>Manufacturer</th>
<th>Furness Controls Ltd.</th>
</tr>
</thead>
<tbody>
<tr>
<td>Type</td>
<td>FC050-WP Ultra-low range differential pressure industrial transmitter</td>
</tr>
<tr>
<td>Electrical supply</td>
<td>240V AC</td>
</tr>
<tr>
<td>Input range</td>
<td>+/- 500mm H₂O</td>
</tr>
<tr>
<td>Output range</td>
<td>+/- 1V DC</td>
</tr>
<tr>
<td>Calibration accuracy</td>
<td>1% FSD</td>
</tr>
<tr>
<td>Linearity</td>
<td>0.5% best straight line</td>
</tr>
<tr>
<td>Hysteresis</td>
<td>0.1 FSD</td>
</tr>
<tr>
<td>Frequency response</td>
<td>20msec - 10sec, fully adjustable</td>
</tr>
<tr>
<td>Volumetric displacement</td>
<td>0.003 cm³ for FSD</td>
</tr>
</tbody>
</table>

A3.2 Calibration

A low-pass R-C filter was installed on the output from each transducer in order to eliminate the 1.7MHz bridge-circuit oscillator frequency.

Tests were carried out to establish that transducer hysteresis and linearity were within satisfactory limits.

The nominal calibration coefficient of 500mm/V was not used though; rather the calibration coefficient was measured for each transducer. This was achieved by applying a range of accurately known pressures to the transducers, logging the signal output, and then fitting a straight line through the data points. The calibration coefficients derived in this way are as follows:

<table>
<thead>
<tr>
<th>Transducer</th>
<th>Calibration coefficient (mm H₂O/Volt)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>496.10</td>
</tr>
<tr>
<td>2</td>
<td>499.15</td>
</tr>
<tr>
<td>3</td>
<td>498.74</td>
</tr>
<tr>
<td>4</td>
<td>497.39</td>
</tr>
<tr>
<td>5</td>
<td>497.34</td>
</tr>
<tr>
<td>6</td>
<td>498.35</td>
</tr>
</tbody>
</table>
APPENDIX 4 COMPUTER HARDWARE

A4.1 Specification

A4.2 Set-up
A4.1 Specification

The data acquisition system is housed in an ADAC 1000 backplane and comprises of the following cards:

- DEC M8186 LSI 11/23 16-bit processor, with memory management and floating point chip.
- DEC M8047 MXV11 multifunction board. Contains two RS232 serial ports, and 16K RAM.
- ADAC 18MP-256 General Purpose Timer. Programmable mode, tick rate etc.
- ADAC 1601 GPT 256K RAM
- ADAC 1012 ADC 12-bit Analogue to Digital Converter. 100KHz throughput.
  Programmable gain. 16 channel multiplexer set up in pseudo-differential mode.
  Relative accuracy 0.035% FSR.
  Inherent quantizing error +/- 1/2 LSB.

A4.2 Set-up

The way in which the LSI 11/23 interfaces with the other hardware is indicated in Fig.A4.1. Initially it was attempted to run the data acquisition system without any filtering of the transducer signals. However it was soon found that the very high oscillator frequencies superimposed on the transducer signals were causing large errors in the ADC readings because of the very small sampling time used by the ADC. Therefore for all testing, low-pass analogue filters were installed.

Considerable effort was expended to ensure that the transducer signals were correctly grounded, since the existence of earth loops can cause significant errors of ADC readings. For further information on ADC grounding considerations, see the ADC manual[125].

The traverse mechanism employs stepper motors which are powered from a purpose-built driver unit (Wray [114]), the control signals for which are generated within the LSI 11/23. Program DRIVE and the associated assembler subroutine PMOVE (Appendix 6) indicate the control logic. Two channels of the High Current Output board are used to interface with the stepper driver unit. One of these channels sets the traverse direction, and the other is used to pulse the motor in individual steps using a square wave format.

A hand-held mouse connected to the LSI 11/23 enables accurate initialisation of probe traverse limits.

At the end of each traverse the test data is transmitted to a remote PDP 11/34 for disc storage. This machine is also used for all subsequent data analysis.
Fig. A4.1 Schematic layout of data acquisition and analysis hardware
APPENDIX 5 FIVE-HOLE PROBES

A5.1 Theory

A5.2 Calibration

A5.3 Using the Five-Hole Probe
This Appendix provides a summary of the pertinent theory of five-hole probes, and also summarises the techniques used for calibration and usage. For a more comprehensive understanding, Wray\cite{117} provides a detailed account of the theory and of the software which has been used to perform the basic manipulation of the data to produce flow velocities and angles. What follows draws heavily on the work reported by Wray.

A5.1 Theory

Since the probe was used exclusively in the non-nulled mode, therefore only this mode is dealt with here. The reasons why this rather than the nulled mode was used are:

i) A substantially simpler traverse system can be employed. Apart from being cheaper this system is required because of the limited access to many of the traverse locations.

ii) Testing time is reduced since there is no time-consuming nulling to be performed.

The convention for hole numbering, and the various angle definitions are given in Fig. A5.1.

Nomenclature additional to that used previously and which is used in this section is as follows:

- $p_n$: pressure sensed by one of the five holes
- $p_i$: pressure sensed by one of the side-holes 2 or 4
- $K_n$: fractional part of flow dynamic head which is sensed by hole $n$
- $X$: pitch parameter
- $Y$: yaw parameter
- $D_p$: dynamic pressure parameter
- $S_p$: stagnation pressure parameter
- PTR: true pitch angle
- PPS: pseudo pitch angle
- YTR: true yaw angle
- YPS: pseudo yaw angle

When the probe is aligned approximately in the flow direction, each of the holes will register a pressure which is the sum of the local static pressure and a proportion of the dynamic head.

$$p_n = p + K_n q$$ \hspace{1cm} A5.1

Since Mach number and Reynolds number effects can be considered negligible, $K_n$ has a value which depends only on the flow direction relative to hole $n$. This relationship is used as the basis for the definitions of $X$, $Y$, $D_p$, $S_p$. The pitch parameter $X$ is defined as:

$$X = \frac{p_1 - p_3}{p_5 - p_i} = \frac{K_1 - K_3}{K_5 - K_1}$$ \hspace{1cm} A5.2
and similarly the yaw parameter $Y$ as:

\[
Y = \frac{P_2 - P_4}{P_2 - P_i} = \frac{K_2 - K_4}{K_5 - K_i}
\]

Since these two parameters have been non-dimensionalised by a function of dynamic head $(P_5 - P_i)$, they are both independent of velocity. For any particular flow direction there is a unique $(X, Y)$ pair provided that the flow angle does not exceed approximately $40^\circ$. The particular hole denoted by subscript $i$ is taken as either hole 2 or 4 depending on which gives the largest value of $(P_5 - P_i)$, in order to maintain the range of $X$ and $Y$ values sensibly low.

For any given flow direction the difference between the pressure sensed by the centre hole and any of the other four must be a function of flow velocity. Using Eqn. A5.1 we obtain:

\[
(P_5 - p_i) = q(K_5 - K_i)
\]

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

\[
q = \frac{P_5 - P_i}{D_p}
\]

Considering incompressible flow along a streamline:

\[
P_t = p + q
\]

Also, considering hole 5, Eqn.A5.1 gives

\[
p = P_5 - K_5 q
\]

Eliminating $p$ from A5.6 and A5.7 gives

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

and non-dimensionalising:

\[
\frac{(P_t - p_5)}{(P_5 - P_i)} = \frac{(1 - K_5)}{(K_5 - K_i)}
\]

The stagnation pressure parameter $S_p$, is defined as:

\[
S_p = \frac{(1 - K_5)}{(K_5 - K_i)}
\]

and therefore we obtain the following expression for stagnation pressure:

\[
P_t = p_5 + S_p(P_5 - p_i)
\]

A5.2 Calibration

The calibration procedure involves subjecting the probe to an airstream of known dynamic
pressure over a range of pitch and yaw angles which cover all variations of flow incidence angles likely to be encountered in service. The calibrations performed on the probes used in this work were all carried out at a flow Mach number of approximately 0.15, which corresponds to typical flow speeds encountered in the test rig. The calibration mechanism is required to rotate the probe in two directions in order to present compound flow incidence onto the probe head. The mechanism used to achieve this is shown in Fig. A7.3. Typical ranges of true yaw and pseudo pitch angle used in the calibrations are from +36° to -36° in increments of 4° or 2°. The calibration procedure is automated in a similar manner to the data acquisition in that it is controlled by an LSI 11/23 microprocessor and the data is transmitted to disc storage on a remote PDP 11/34. The data contained in the calibration file consists of PPS, YTR, Dp, Sp for each X,Y pair.

A5.3 Using the Five-Hole Probe

When test data has been acquired the basic steps which have to be followed in order to derive velocities are as follows:

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

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

iii) A least squares bi-quadratic surface is fitted to each of the four parameter arrays, and the value at the test X,Y coordinates calculated.

iv) Using these parameter values, and also p5,pi; the true pitch, true yaw, stagnation pressure and dynamic pressure are calculated, using the above equations.

v) From these quantities it is relatively simple to calculate any required incidences and velocity components.

The software used to perform the above analytical steps is complex and is described in greater detail by Wray[117]. When the data is output from this software, it is in a generalised form and consists of data files listing coordinate values, angles, velocities and pressures. In order to present the data in the form shown in Chapter 4, a range of additional programs have to be run, with each different form of presentation requiring a different program to be run.
a) Hole numbering

b) Flow angles and velocity components

- PTR - True pitch angle
- PPS - Pseudo pitch angle
- YTR - True yaw angle
- YPS - Pseudo yaw angle

Fig. A5.1 Five hole probe labelling conventions
APPENDIX 6 COMPUTER PROGRAMS

Program DRIVE
Subroutine PMOVE
Program BPITCH
Program BRAVA
Program DRIVE is used to drive a DISA 52C01 stepper motor via a purpose-built (in department) driver box, from the HCO (high current output) board in the LSI, and to acquire data from probe(s). A variety of probes can be used, though only one type per run. 5 ADC channels have been allotted (this can easily be increased) for probe readings, so could use e.g. up to 5 pitot probes, or 1 five-hole probe, etc. Channels 0 and 1 of the HCO are used, for no. of probe increments and direction respectively.

Program DRIVE performs each traverse in two halves; the first with the probe moving outwards from the I/W to approx. mid-annulus; the second with the probe moving inwards from the O/W to approx. mid-annulus.

ADC channels should be connected as follows:
0 - Comark 6110 monitoring inlet temperature
1 - probe transducer
2 - probe (if req'd) " 2
3 - " " 3
4 - " " 4
5 - probe(5HP), or POW-PW2 " 5
6 - P (lab) - P (W2) " 6

The FORTRAN main segment calls an assembler subroutine to move the probe by the stepper motor and then to wait for the probe reading to settle. The 1601 GPT programmable clock is used throughout, i.e. SWITCH RTC OFF. Library subroutines are called to read the ADC and to send the data downline to the PDP 11/34.

The data is arranged in a one-dimensional real array, the first eight words of which have the following significance:

<table>
<thead>
<tr>
<th>Word</th>
<th>Meaning</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>no. of points in traverse</td>
</tr>
<tr>
<td>2</td>
<td>rotor rpm for traverse</td>
</tr>
<tr>
<td>3</td>
<td>ratio of inlet static (std/test)</td>
</tr>
<tr>
<td>4</td>
<td>(Poffset - Pref.static)</td>
</tr>
<tr>
<td>5</td>
<td>no. of columns in 2-d array form</td>
</tr>
<tr>
<td>6</td>
<td>inner wall radius</td>
</tr>
<tr>
<td>7</td>
<td>outer wall radius</td>
</tr>
<tr>
<td>8</td>
<td>(Poffset - Po.w.static) (pitot only)</td>
</tr>
</tbody>
</table>

The rest of the data, when it is subsequently processed can be rearranged to form a two-dimensional array with columns arranged as follows:

<table>
<thead>
<tr>
<th>Column</th>
<th>Quantity</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>radius (mm)</td>
</tr>
<tr>
<td>2</td>
<td>angular position (rads)</td>
</tr>
<tr>
<td>3</td>
<td>thermocouple meter output (volts)</td>
</tr>
<tr>
<td>4</td>
<td>probe reading (volts)</td>
</tr>
</tbody>
</table>
PROGRAM DRIVE

PARAMETER (PI=3.14159, NSTART=0, RADIW=379.98, PSTD=101325.0)
PARAMETER (TABS=273.15, DMCONV=9.8088)
CHARACTER*1 DUM, FILNAM*10, PTYPE*3
DIMENSION IGN(2), SET(8), SL(6)
DIMENSION IGAIN(8), DAT(410), NINC(3)
DATA DAT(1)/42.0/
DATA IGN(1), IGN(2)/2,2/
DATA SL(1), SL(2), SL(3)/496.10, 499.15, 498.74/
DATA SL(4), SL(5), SL(6)/497.39, 497.34, 498.35/
COMMON /PMOVE/ NPINC, NWAIT, NMOT, NRATE

C ------------ Read required variables
C ------------
70 WRITE(5,100)'Probe type (PIT/5HP)'
READ(5,’(A)’) PTYPE
IF (PTYPE.NE.'5HP'.AND.PTYPE.NE.'PIT') GOTO 70
WRITE(5,100)'Default parameters? [Y/N]'
READ(5,’(A)’) DUM
IF (DUM.EQ.'N') THEN
  WRITE(5,100)'No. of probes being driven'
  READ(5,*) NOPRO
  WRITE(5,100)'Probe settling time (1/10 sec)'
  READ(5,*') IWAIT
  WRITE(5,100)'No. of samples at each point'
  READ(5,*) NSAMP
  WRITE(5,100)'Interval between samples (msec)'
  READ(5,*') NINT
  WRITE(5,100)'No. of msec per step'
  READ(5,*') NRATE
  WRITE(5,100)'Typical lab. temp. (deg. C)'
  READ(5,*') TTYP
ELSE
  NOPRO=1
  IWAIT=100
  IF (PTYPE.EQ.'PIT') IWAIT=25
  NSAMP=250
  NINT=20
  TTYP=25.0
  FORMAT(5,130) PTYPE, NOPRO, IWAIT, NSAMP, NINT, TTYP
ENDIF
WRITE(5,100)'Annulus height (mm)'
READ(5,*) ANNHT
WRITE(5,100)'Rotor RPM'
READ(5,*') RPM
WRITE(5,100)'No. of O.G.V.s'
READ(5,*) NBLADE
C Set up AVGADC, SNDDAT arguments
C------------------------------------------------------
SMPFRQ=1000./FLOAT(NINT)
DO 80 II=1,8
     IGAIN(II)=10
IF (PTYPE.EQ.'PIT') THEN
     NCHAN=(1*NOPRO)+3
     PDIAM=0.9144
ENDIF
IF (PTYPE.EQ.'5HP') THEN
     NCHAN=(5*NOPRO)+3
     PDIAM=1.727
ENDIF
C------------------------------------------------------
C Set up PMOVE arguments, various other parameters
C------------------------------------------------------
NMOT=1
TTYP=TABS+TTYP
RDRT=RPM/SQRT(TTYP)
!Rig operating point
BSPACE=2.0*PI/FLOAT(NBLADE)
NCOL=NCHAN
NPT=DAT(1)
NPTS=NPT+7/NCOL+1
DAT(5)=NCOL
DAT(6)=RADIW
DAT(7)=RADIW+ANNHT
ANNHT=ANNHT-PDIAM
INC995=INT(ANNHT*0.995*48.0)
!Motor has 48 steps per rev
PMOV=ANNHT/98.0
!198 is 2*(7 + 2*7 + 4*7)
NSINC=INT(PMOV*48.0)
NINC(1)=NSINC
NINC(2)=NSINC*2
NINC(3)=NSINC*4
RADINN=RADIW+PDIAM/2.0
RADOUT=RADINN+ANNHT
IFLAG=0
C Measure initial inlet temperature
C------------------------------------------------------
CALL AVGADC(0,8,100,IGAIN,10.0,SET)
TLAB=SET(1)*100.0+TABS  !100.0 is cal. for COMARK 6110
PRMW=SET(6)*SL(5)*-1.0
PLMR=SET(8)*SL(6)
C Start loop for traversing
C------------------------------------------------------
IHALLF=0
SUM1=0.0
SUM2=0.0
SUM3=0.0
NSET=INT(RDRT*SQRT(TLAB))
WRITE(5,100) 'Angular position [REAL]'
READ(5,*) THETA
THETA=THETA*BSPACE
WRITE(5,100) 'Ambient air pressure (mBar)'
READ(5,*) PLAB
RHO=(100.0*PLAB-PLMR*DMCONV)/287.1/TLAB
PRAT=GSTD/(100.0*PLAB-PLMR*DMCONV)
DAT(2)=FLOAT(NSET)
DAT(3)=PRAT
IF (PTYPE.EQ.'PIT') DAT(8)=PRMW*PRAT/25.4
IF (IFLAG.NE.1) THEN
  WRITE(5,100) (Poffset - Pref.plane) (inH2O)
  READ(5,*) DAT(4)
  DAT(4)=DAT(4)*PRAT
  IF (DAT(4).EQ.0.0) IFLAG=1
ENDIF
WRITE(5,30) PLMR,PRAT,TLAB,NSET,RHO
30 FORMAT (/2X, 'Plab - Pw2',T42,F6.1,' mm H2O',
     1/2X, 'Ambient pressure ratio',T42,F8.5,
     2/2X, 'Inlet temperature',T43,F5.1, ' deg. C',
     3/2X, 'Set rotor speed to',T42,I5, ' RPM',
     4/2X, 'Air density at ref. plane',T42,F7.4, ' kg/m3')
C ------------------------------
C Halt while probe is set up
C ------------------------------
WRITE(5,100) 'Set OGV. Do you want to initialise probe?'
READ(5,'(A)') DUM
IF (DUM.EQ. 'Y') THEN
  WRITE(5,100) 'Set up probe at I/W; C/R when ready'
  READ(5,'(A)') DUM
  WRITE(5,'(A)') 'Probe initialisation in progress'
  IPOS=0
  NPINC=INC995
  NWAIT=2
  IPOS=IPOS+NPINC
  CALL PMOVE
CALL AVGADC(14,2,100,IGN,1000.0,SET)
IF (SET(1).LT.0.2) THEN
  NPINC=0
  IF (SET(2).GT.0.2) NPINC=-1
  IF (SET(2).GT.0.6) NPINC=1
  IPOS=IPOS+NPINC
  CALL PMOVE
  GOTO 10
ENDIF
INC100=IPOS
ENDIF
NPINC=-IPOS
IPOS=IPOS+NPINC
NWAIT=IWAIT
CALL PMOVE
C ------------------------------
C Set up table for display of measured values
C ------------------------------
WRITE(5,60) (K-1,K=1,NCHAN-2)
60 FORMAT(1H ,//,2X, 'NO.',5X, 'RAD',6X, 'ANG',6(3X,1L, 'CHAN'))
C ------------------------------
C Start automated traverse
C ------------------------------
DO 50 LC=1,NPT
  IF (LC.EQ. (NPT/2+1)) THEN
    NPINC=INC100-IPOS
    IPOS=IPOS+NPINC
    CALL PMOVE
  ENDIF
50
IHALF=1
ENDIF
IF (IHALF.EQ.0) THEN
  J= (LC-1)*NCOL+9
  IMOVE=0
  IJ=1
  IF (LC.GT.NPT/6) IJ=2
  IF (LC.GT.NPT/3) IJ=3
  NPINC=NINC(IJ)
  NTINC=NTINC+NPINC
  IF (LC.EQ.1) THEN
    NPINC=0
    NTINC=0
  ENDIF
ELSE
  J=NPT*NCOL+9-(LC-NPT/2)*NCOL
  IMOVE=0
  IJ=1
  IF (LC.GT.2*NPT/3) IJ=2
  IF (LC.GT.5*NPT/6) IJ=3
  NPINC=NINC(IJ)
  NTINC=NTINC+NPINC
  IF (LC.EQ.NPT/2+1) THEN
    NPINC=0
    NTINC=0
  ENDIF
  NPINC=-NPINC
ENDIF
DAT(J+1)=THETA
C ---------------------------------------------C
Move probe and wait for reading to settle
C ---------------------------------------------
IPOS=IPOS+NPINC
CALL PMOVE
DAT(J)=RADINN+(FLOAT(IPOS)/48.0) !Probe tip radius
C Read pressure probe/hot wire measurements
C CALL AVGADC (NSTART, 8, NSAMP, IGAIN, SMPFRQ, SET)
WRITE (5, (40)) (LC, (DAT(I), II=J, J+1), (SET(I), II=1, NCHAN-2)
40 FORMAT(1H ,X,I3,2X,F8.2,F8.5,<NCOL-1>F8.4)
SUM1=SUM1+SET(1)
SUM2=SUM2+SET(8)
SUM3=SUM3-SET(6)
DO 140 K=1,NCHAN-2
  KI=K+J+1
  IF (K.EQ.1) THEN
    DAT(KI)=SET(K)
  ELSE
    DAT(KI)=SET(K)*SL(K-1)*PRAT
  ENDIF
140 CONTINUE
50 CONTINUE
C Send data to PDP 11/34
C WRITE(5,100)'Enter PDP filename(will be .DAC;1)'
READ(5,'(A)')FILNAM
CALL SNDDAT (DAT, FILNAM, NPTS, NCOL)

TLAB = SUM1 * 100.0 / DAT(1) + TABS
PLMR = SUM2 * SL(6) / DAT(1)
PRMW = SUM3 * SL(5) / DAT(1)
GOTO 20

C
100  FORMAT (/2X, A, T50, ', ', $)
END
SUBROUTINE PMOVE

Author : K.F. Young

.TITLE PMOVE
.LIST BEX,BIN
.ENABL LC,AMA
.PSECT PMOVE,RW,D,OVR,REL,GBL

NPINC: 0
NWAIT: 0
NMOT: 0
NRATE: 0

.PSECT

CSR1=542
CSR2=550
CLKST=176760
CLKBF=176762
CLKV1=370
CLKV2=372
HCOBF=164000

DBIT: 0
IBIT: 0

PMOVE:

; Test to see which stepper motor to use, and calculate the relevant bits of the HCO status word

MOV NMOT,R0
MOV #1,R1
DEC R0
BEQ 7$
ASL R0

6$:
ASL R1
DEC R0
BGT 6$

7$:
MOV R1,IBIT
ASL R1
MOV R1,DBIT
CLR R0
CLR R1

MOV NPINC,R2
BEQ FLAG
BPL 1$
NEG NPINC
BIC DBIT,#HCOBF
BR 2$

1$:
BIS DBIT,#HCOBF

2$:
MOV #ISR1,#CLKV1
MOV #340,#CLKV2
MOV NPINC,R0
ASL R0
CLR R1
MOV NRATE,#CLKBF
MOV #CSR1,#CLKST

; Clock vector
; Interrupt priority
; Interrupt count
; Multiply by 2
; Set 'switch' to on
; Load count buffer
; Set clock status
INC @#CLKST ;Start clock

;HOLD:
WAIT
TST R0 ;Finished move yet?
BGT HOLD ;No, loop again
BIC #1,@#CLKST ;Yes, stop clock

;FLAG:
MOV #ISR2,@#CLKV1 ;Reset clock vector
MOV NWAIT,R3 ;Wait time, 1/10 sec
BEQ 5$ ;Don't wait if zero
MUL #10.,R3 ;Convert to ticks
MOV R3,@#CLKBF ;Load count buffer
MOV #CSR2,@#CLKST ;Set clock status
INC @#CLKST ;Start clock
WAIT
BIC #1,@#CLKST ;Stop clock

5$:
BIC DBIT,@#HCOBF ;Leave dir'n bit low
RTS PC

;ISR1:
TST R1 ;Decide whether to
BEQ 3$ ;switch HCO on or off
BIC IBIT,@#HCOBF ;HCO off
BR 4$

3$:
BIS IBIT,@#HCOBF ;HCO on

4$:
DEC RO ;Decrement int. count
INC R1 ;Set alternate on/off
BIC #2,R1 ;flag
BIS #200,@#CLKST ;Reset OVFL bit
RTI

;ISR2:
RTI

; .END
Program BPITCH produces a radial distribution of pitch-
averaged area-weighted non-dimensional axial velocity. An o/p
file from B5HP3 or BPITO(S) is read in, the wall values of
velocity being calculated. The data is interpolated radially
to a regular grid, then a circumferential interpolation is
performed over one blade space at each radial position to
produce the required velocity array. This array is then output
to a .DAC file suitable for plotting with DATS DISPLY.

Program BPITCH

PARAMETER AMIN=0.0,PI=3.14159,NGC=50, SZ=50, NRREQ=50
PARAMETER NRR2=500
CHARACTER*13 FILIN, FILE*30, PTYP*3, FLAG1*1
DIMENSION ANG(SZ), V(SZ, SZ), SVA(SZ), AVER(SZ)
DIMENSION V1(SZ), RAD(SZ), RAD2(SZ), RAD3(SZ)
DIMENSION RHO(SZ), TAU(SZ)

WRITE(5, 100) 'Name for input file'
READ (5, , (A)) FILIN
WRITE (5, 100) 'Probe type (PIT/5HP)'
READ (5, , (A)) PTYP
WRITE (5, 100) 'No. of O.G.V.s'
READ (5, *) BLADE
WRITE (5, 100) 'Area-weight radially? [Y/N]'
READ (5, , (A)) FLAG1
WRITE (5, 100) 'Name of .DAC file for plot'
READ (5, , (Q, A)) 'NCHR, FILE
FILE(NCHR+1:)=' .DAC'
WRITE (5, 100) 'Symbol number for this plot'
READ (5, *) NSYM

WRITE(5, 100) 'Name for input file'
READ(5, , (A)) FILIN
WRITE(5, 100) 'Probe type (PIT/5HP)'
READ(5, , (A)) PTYP
WRITE(5, 100) 'No. of O.G.V.s'
READ(5, *) BLADE
WRITE(5, 100) 'Area-weight radially? [Y/N]'
READ(5, , (A)) FLAG1
WRITE(5, 100) 'Name of .DAC file for plot'
READ(5, , (Q, A)) 'NCHR, FILE
FILE(NCHR+1:)=' .DAC'
WRITE(5, 100) 'Symbol number for this plot'
READ(5, *) NSYM

Open( UNIT=1, NAME=FILIN, TYPE='OLD', READONLY) READ(1, *) NRAD, NCIRC, D, RADIW, RADOW NRAD=NRAD+2 IF (PTYP.EQ.'5HP') THEN READ(1, *) (RAD(I), ANG(J), D, D, V(I, J), D, D, D, J=1, NCIRC), 1 I=2, NRAD-1 ELSE IF (PTYP.EQ.'PIT') THEN READ(1, *) (RAD(I), ANG(J), D, D, D, V(I, J), J=1, NCIRC), 1 I=2, NRAD-1 ELSE GOTO 9999 ENDIF
CLOSE(UNIT=1)
C---------------------------------
C Set 'wall values' of velocity
C---------------------------------
R3=RAD(2)
R4=RAD(NRAD-1)
ANNHT=RADOW-RADIW
RAD(1)=RADIW
RAD(NRAD)=RADOW
RFI=(RAD(1)-RAD(2))/(RAD(3)-RAD(2))
RFO=(RAD(NRAD)-RAD(NRAD-1))/(RAD(NRAD-1)-RAD(NRAD-2))
DO 10 J=1,NCIRC
   V(1,J)=(V(3,J)-V(2,J))*RF+V(2,J)
   IF(V(1,J).GT.V(2,J))V(1,J)=V(2,J)
   IF(V(1,J).LT.0.0)V(1,J)=0.0
V(NRAD,J)=(V(NRAD-1,J)-V(NRAD-2,J))*RFO+V(NRAD-1,J)
   IF(V(NRAD,J).GT.V(NRAD-1,J))V(NRAD,J)=V(NRAD-1,J)
   IF(V(NRAD,J).LT.0.0)V(NRAD,J)=0.0
10 CONTINUE
C ----------------------------------

C Radial interpolation

DR=ANNHT/FLOAT(NRREQ-1)
DO 60 J=1,NCIRC
   DO 70 I=1,NRAD
      VI(I)=V(I,J)
   CONTINUE
   CALL SPOCOEF(NRAD,RAD,VI,SVA,RHO,TAU)
   DO 80 I=1,NRREQ
      RREQ=DR*FLOAT(I-1)+RADIW
      CALL SPLINE(NRAD,RAD,VI,SVA,RREQ,ANS)
      V(I,J)=ANS
      RAD2(I)=RREQ
   CONTINUE
CONTINUE
60 CONTINUE
C Perform circumferential interpolation at each radial
C position, over a single blade space.

K=NRREQ/2
AVDA=PI*(RAD2(K+1)**2-RAD2(K)**2)/BLADE
BSPACE=2.0*PI/BLADE
DELA=BSPACE/FLOAT(NGC-1)
VMAX=0.0
DO 500 I=1,NRREQ-1
   R1=RAD2(I)
   R2=RAD2(I+1)
   DA=PI*(R2**2-R1**2)/BLADE
   DAR=DA
   IF(FLAG1.EQ.'Y')DAR=AVDA
   IF(NCIRC.EQ.1)THEN
      AVER(I)=(V(I,1)+V(I+1,1))/2.0*DA/DAR
      GOTO 90
   ENDIF
   DDA=DA/FLOAT(NGC-1)
   DO 20 J=1,NCIRC
      VI(J)=(V(I,J)+V(I+1,J))/2.0
   CONTINUE
CALL SPCOEF (NCIRC, ANG, V1, SVA, RHO, TAU)
SUM1=0.0
AREQ=AMIN+DELA/2.0
DO 30 J=1,NGC-1
    CALL SPLINE (NCIRC, ANG, V1, SVA, AREQ, ANS)
    IF (ANS.LT.0.0) ANS=0.0
    SUM1=SUM1+ANS*DDA
    AREQ=AREQ+DELA
30 CONTINUE
AVER(I)=SUM1/DAR
90 RAD3(I)=(R1+R2)/2.0
IF (AVER(I).GT.VMAX) THEN
    VMAX=AVER(I)
    NFLG=I
ENDIF
500 CONTINUE
C -----------------------
C Find b/l parameters
C -----------------------
120 SUM1=0.0
SUM2=0.0
DO 120 I=1,NRREQ-1
    RAT=AVER(I)/VMAX
    SUM1=SUM1+(1.0-RAT)*DR
    SUM2=SUM2+(1.0-RAT)*DR*RAT
    IF (I.EQ.NFLG) THEN
        DELI=SUM1
        SUM1=0.0
        THEI=SUM2
        SUM2=0.0
    END IF
120 CONTINUE
HINN=DELI/THEI
DELIP=DELI*100.0/ANNHT
THEIP=THEI*100.0/ANNHT
DELO=SUM1
THEO=SUM2
HOUT=DELO/THEO
DELOP=DELO*100.0/ANNHT
THEOP=THEO*100.0/ANNHT
OPEN (UNIT=4, NAME='BPITCH.LST', TYPE='NEW')
WRITE (4, '(2X,36(""))')
WRITE (4, '(2X,A,A,/2X,A,F4.0,/2X,A,A)')
1'Input file : ',FILIN,' Probe type : ',PTYPE,
2'No. blades : ',BLADE,' Radial area wt : ',FLAG1
WRITE (4, 130) DELI, DELIP, THEI, THEIP, HINN
FORMAT (//10X, 'Boundary layer parameters',
1//2X, 'DELTA STAR (inner wall) ',T35,F6.1,' mm',
2'(',F6.1,' % ann ht.)',
3//2X, 'THETA (""),',T35,F6.1,' mm',
4'(',F6.1,' % ann ht.)',
5//2X, 'Shape parameter (""),',T35,F6.1)
WRITE (4,140) DELO, DELOP, THEO, THEOP, HOUT
140 FORMAT (/2X, 'DELTA STAR (outer wall) ',T35,F6.1,' mm',
1'(',F6.1,' % ann ht.)',
2//2X, 'THETA (""),',T35,F6.1,' mm',
3'(',F6.1,' % ann ht.)',
4//2X, 'Shape parameter (""),',T35,F6.1)
WRITE(4,'(2X,36(''*''))')
CLOSE(UNIT=4)

C -------------------------------
C Second radial interpolation, to bring data within 
C radial range tested over. Output results to file.
C -------------------------------

DR=(R4-R3)/FLOAT(NRR2-1)
CALL PLTST(FILE,1.0,1.3,0.769,1)
CALL SPCOEF(NRREQ-1,RAD3,AVER,SVA,RHO,TAU)
K=2
DO 40 I=1,NRR2
   RREQ=DR*FLOAT(I-1)+R3
   CALL SPLINE(NRREQ-1,RAD3,AVER,SVA,RREQ,ANS)
   NUD=1
   LSYM=0
   IF (I.EQ.1) NUD=0
   IF (ABS(RREQ-RAD(K)).LE.(DR/2.0)) THEN
      LSYM=NSYM
      K=K+1
   ENDIF
   RREQ=(RREQ-RADIW)/ANNHT
   CALL GRAPLT(RREQ,ANS,NUD,LSYM)
40 CONTINUE
CALL PLTFIN

C 100 FORMAT(/2X,A,T50,' : ',$)
9999 STOP
END
Program BRAVA calculates the radial distribution of pitch-averaged velocity component or angle. If an angle is selected it can be either the pitch or the yaw angle. If a velocity component is selected it can be in either the radial or the circumferential direction.

An output file from B5HP2 is read in, one traverse at a time for a single blade space. For each circumferential traverse the required velocity at each point is calculated, then averaged for that traverse. An interpolation is then performed to present this data at regular radial positions. PLTLIB2 is used to output the data to a .DAC file for plotting.

PROGRAM BRAVA
PARAMETER (PI=3.14159,DRCONV=0.0174532,THMIN=0.0)
PARAMETER (NSZ=41,SZ1=50,NRREQ=500)
DIMENSION THETA(NSZ),PI1(NSZ),YAI(NSZ),CP(NSZ),PLOC(NSZ)
DIMENSION V1(SZ1),RAD(SZ1),P5(NSZ)
DIMENSION SVE(SZ1),RHO(SZ1),TAU(SZ1)
CHARACTER*30 FILIN,FILOUT,FLAG*1,FLAG2*1

WRITE (5,100) 'Input file name from B5HP2'
READ (5,*(A))FILIN
WRITE (5,100) 'No. of O.G.V.s in ring'
READ (5,*)BLADE
WRITE (5,100) 'Rad vel/Circ vel/Pitch ang/Yaw ang [R/C/P/Y]'
READ (5,*(A))FLAG
WRITE (5,100) 'Mass-weight this parameter? [Y/N]'
READ (5,*(A))FLAG2
WRITE (5,100) 'Mean velocity from BDIAN(S) (m/s)'
READ (5,*(A))VBAR
WRITE (5,100) 'Name for output file'
READ (5,'(Q,A)')NCHR,FILOUT
FILOUT(NCHR+1:()='DAC'
WRITE (5,100) 'Symbol number for this plot'
READ (5,*(A))NSYM

---

BSPACE=2.0*PI/BLADE
OPEN(2,NAME=FILIN,TYPE='OLD',READONLY)
READ (2,*)NRAD,NCIRC,RIN,ROUT
ANNHT=ROUT-RIN
DO 300 I=1,NRAD
NCREQ=NCIRC
NFLG=0
DO 301 K=1,NCIRC
  IF (NFLG.EQ.1) THEN

READ (2,*) D, D, D, D, D, D, D, D
GOTO 301
ENDIF
READ (2,*) RAD (I), THETA (K), PI1 (K), YA1 (K),
CP (K), SP, PLOC (K), P5 (K)
IF (THETA (K) .GT. BSPACE) THEN
  NCREQ = K - 1
  NFLG = 1
ENDIF
301 CONTINUE

C Calculate required velocity component at
each point and pitch-average
C--------------------------------------------------------
SUM1 = 0.0
SUM2 = 0.0
DO 30 J = 1, NCREQ
  IF (PI1 (J) .EQ. 123.4) GOTO 30
  IF (CP (J) .EQ. 0.0) GOTO 30
  CHECK = (16.0 * (P5 (J) - PLOC (J)) / CP (J))
  IF (CHECK .LT. 0.0) GOTO 30
  V = SQRT (CHECK)
  PPS = PI1 (J) * DRCONV
  YTR = YA1 (J) * DRCONV
  PTR = ATAN (TAN (PPS) / COS (YTR)) / DRCONV
  VR = V * SIN (PPS)
  VC = V * COS (PPS) * SIN (YTR)
  VAX = V * COS (PPS) * COS (PRANG - YTR)
  WT = 1.0
  IF (FLAG2 .EQ. 'Y') WT = VAX
  IF (FLAG .EQ. 'R') VP = VR / VBAR !Radial velocity component
  IF (FLAG .EQ. 'C') VP = VC / VBAR !Circ. velocity component
  IF (FLAG .EQ. 'P') VP = PTR * WT !True pitch angle
  IF (FLAG .EQ. 'Y') VP = YA1 (J) * WT !True yaw angle
  SUM1 = SUM1 + VP
  SUM2 = SUM2 + VAX
30 CONTINUE
V1 (I) = SUM1 / FLOAT (NCREQ)
IF (FLAG2 .EQ. 'Y') V1 (I) = SUM1 / SUM2
300 CONTINUE
CLOSE (2)

C Interpolate to regular radial positions
C and output results to .DAC file for DATS
C--------------------------------------------------------
YR = 0.4
IF (FLAG .EQ. 'P' .OR. FLAG .EQ. 'Y') YR = 40.0
CALL PLTST (FILOUT, 1.0, YR, 0.625, 1)
R1 = RAD (1)
DR = (RAD (NRAD) - R1) / FLOAT (NRREQ - 1)
CALL SPcoef (NRAD, RAD, V1, SVE, RHO, TAU)
K = 1
DO 20 I = 1, NRREQ
  RREQ = R1 + DR * FLOAT (I - 1)
  RND = (RREQ - RIN) / ANNH T
  CALL SPLINE (NRAD, RAD, V1, SVE, RREQ, UREQ)
  LSYM = 0
  IF (ABS (RREQ - RAD (K)) .LE. (DR / 2.0)) THEN

309
LSYM=NSYM
K=K+1
ENDIF
NUD=1
IF (I.EQ.1) NUD=0
CALL GRPLT(RND,UREQ,NUD,LSYM)
20 CONTINUE
CALL PLTFIN
C
100 FORMAT (/2X,A,T50,':','$,)
STOP
END.
APPENDIX 7 PHOTOGRAPHS

A7.1 The Test Rig

A7.2 Straight and Chevron Stator Blades

A7.3 Five-Hole Probe Calibration Rig

A7.4 Inter-Blade Five-Hole Probe and Associated Traverse Mechanism

A7.5 Radial Traverse Gear

A7.6 Circumferential Movement of OGV Ring
Fig.A7.1 The test rig
Fig. A7.2a Chevron stator blades
Fig.A7.2b Straight stator blades
Fig. A7.3 Five-hole probe calibration rig
Fig.A7.4 Inter-blade five-hole probe and associated traverse mechanism
Fig.A7.5 Radial traverse gear
Fig. A7.6 Circumferential movement of OGV ring