Evaluation of Swirl and Tabs in Short Annular Diffusers

by

David J. Cerantola
B.A.Sc., M.A.Sc.

A thesis submitted to the
Department of Mechanical and Materials Engineering
in conformity with the requirements for
the degree of Doctor of Philosophy

Queen’s University
Kingston, Ontario, Canada
May 2014

Copyright © David J. Cerantola, 2014
I hereby declare that I am the sole author of this thesis.

I authorize Queen’s University to lend this thesis to other institutions or individuals for the purpose of scholarly research.

I further authorize Queen’s University to reproduce this thesis by photocopying or by other means, in total or in part, at the request of other institutions or individuals for the purpose of scholarly research.
Abstract

Short annular diffusers were essential components for turbomachines that have been used to expand the air entering the compressor, as interstage ducts between gas generators and power turbines, and on the exhaust gases exiting the turbine. The industrial community was interested and invested in improving diffuser design that was challenging owing to the unfavourable fluid flow effects. Efficient design of fluid flow devices was possible through the complementary use of experimental testing and computational fluid dynamics (CFD). A numerical shape optimization study was undertaken to determine preferential annular diffuser configurations. Experimental data were compared against CFD that simulated the steady-state Reynolds-averaged Navier-Stokes equations with two-equation turbulence models.

This investigation reached equivalent conclusions with respect to the influences associated with diffuser geometry and swirl. Vorticity effects caused by square tabs, that were not as well understood, were investigated. The tabs were effective in reducing the central toroidal recirculation zone created by a swirling flow, but at a static pressure penalty for the area ratio, $AR \leq 2.73$, diffusers tested. Results identified several shortcomings in the CFD that typically over-estimated pressure recovery and outlet velocity uniformity; however, properly qualitatively predicted wall pressure distributions and outlet velocity profiles. The use of CFD on modest grids, with preference given to the realizable $k$-$\epsilon$ turbulence model, for annular diffusers that have length to inlet height ratio of 12 and at least $AR = 2.73$ with up to $20^\circ$ inlet swirl was encouraged as a design tool.
Sitting atop a great green hill
Looking out at the cloudy skies and luscious green pastures
Rays of sunlight struggling to twinkle off the spires in the distance
Wondering what would have happened if two more flowers were picked
or how history would have changed over one metre
Could it happen?

Would it have happened?
Ancestors have had revolutionary revolutions here.
But it is just a mound to me.

Blame mead!
Acknowledgments

For some, the motivation for pursuing higher education is to obtain another piece of paper as quickly as possible. I, however, was keen on writing papers and attending conferences, so was fortunate to visit Orlando, Copenhagen, San Antonio, and Düsseldorf. Each conference was rewarding in its own way that provided directional insight for my research and the opportunity to gain worldly travel experience. Dr. Birk’s encouragement and financial support made attending so many conferences possible.

As the laggard in a research group of colleagues, including Billy Ng, that worked on similar projects, I had the benefit of implementing their methods and acquiring inspiration from their efforts. In particular, special thanks was extended to James Crawford for participating in informative conversations and assistance in the machine shop. Acknowledgement was also given to Dave Adams for sharing the lab space and manufacturing the centre-bodies using single point incremental forming: his persistence and creativity were essential for forming parts with steep wall angles.

Funding provided by W. R. Davis Engineering Ltd. and the Natural Sciences and Engineering Research Council of Canada under the Collaborative Research Development grant, and the Seanix Graduate Award was appreciated.
Table of Contents

Abstract ................................................................................................................ iii
Acknowledgments .................................................................................................... v
Table of Contents .................................................................................................... vi
List of Tables .......................................................................................................... xi
List of Figures ......................................................................................................... xiii
Nomenclature .......................................................................................................... xix

Chapter 1 Introduction ......................................................... 1
  1.1 Objective ........................................................................................................... 3
  1.2 Contributions .................................................................................................... 3
  1.3 Scope ................................................................................................................ 4

Chapter 2 Theory and Literature Review .................................................... 5
  2.1 Performance Parameters ............................................................................... 6
    2.1.1 Averaging .................................................................................................. 9
  2.2 Theory ............................................................................................................. 9
    2.2.1 Turbulence ............................................................................................... 10
    2.2.2 Compressibility ....................................................................................... 11
    2.2.3 Aerodynamic Blockage .......................................................................... 12
    2.2.4 Swirl .......................................................................................................... 13
    2.2.5 Diffusion ................................................................................................... 17
    2.2.6 Curvature ................................................................................................ 20
    2.2.7 Struts ........................................................................................................ 21
  2.3 Computational Fluid Dynamics ........................................................................ 21
    2.3.1 Turbulence Models .................................................................................. 24
6.3.2 Tab Wake ............................................. 92
6.3.3 Number of Tabs Experimental Study ............................................. 96
6.3.4 Number of Tabs CFD Study ............................................. 99
6.3.5 Amount of Swirl ............................................. 100
6.3.6 Comparison to Experiment ............................................. 102

Chapter 7 Conclusions and Recommendations .............................. 110
7.1 Conclusions ............................................. 110
7.1.1 CFD Results Summary ............................................. 111
7.2 Design Advice ............................................. 112
7.3 Benefits for Industrial Sponsor ............................................. 113

References ............................................................................. 115

Appendix A Experimental Methods Supplementary ......................... 125
A.1 Swirler Profiles ............................................. 125
A.2 Three Hole Probe Reduction ............................................. 126
A.3 Pressure Transducer Sampling Analysis ............................................. 130
A.4 Measurement Uncertainty ............................................. 133
  A.4.1 Uncertainty Propagation ............................................. 135
A.5 Bias Assessment ............................................. 137
  A.5.1 Inlet Traverse Analysis ............................................. 138
  A.5.2 Outlet Traverse Analysis ............................................. 140

Appendix B Geometry Optimization ............................................. 146
B.1 Literature Review ............................................. 146
B.2 Problem Description ............................................. 147
  B.2.1 Constraints ............................................. 149
B.3 Methodology ............................................. 150
B.4 Computational Verification ............................................. 153
B.5 Optimization Study Results ............................................. 155
Appendix C  CFD Executable Scripts

C.1 Geometry Input File ....................................................... 160
C.2 Boundary Condition Input File ........................................ 160
C.3 Linux Bash Scripts .......................................................... 162
C.4 Mesh Generation Script .................................................. 163
C.5 CFD Journal Script ......................................................... 173
C.6 Fortran Post-Processing Script ......................................... 177

Appendix D  Part Drawings ..................................................... 180

Appendix E  Diffuser Geometry Study Supplementary .................. 191

E.1 Literature Review ............................................................ 191
E.2 Experimental Results ...................................................... 193
  E.2.1 Inlet ........................................................................ 193
  E.2.2 Wall ........................................................................ 195
  E.2.3 Outlet ..................................................................... 196
  E.2.4 Reynolds Number Influence ....................................... 198
  E.2.5 Mass Flow Comparison ............................................ 200
  E.2.6 Performance ............................................................ 201
  E.2.7 Centre-Body Importance ........................................... 202
  E.2.8 Diffuser Effectiveness ............................................... 203
  E.2.9 Diffuser Sections ...................................................... 204
E.3 Numerical Results ............................................................ 205
  E.3.1 Residual Convergence ............................................... 205
  E.3.2 Coarse Grid Resolution ............................................. 207
  E.3.3 Reynolds Number Effects ......................................... 208
  E.3.4 Turbulence Quantity Inputs ....................................... 210
  E.3.5 Centre-Body Curvature ............................................. 213
  E.3.6 Comparison to Experiment ....................................... 215
E.4 Conclusions ................................................................. 222
List of Tables

2.1 Tested augmentation device review geometries ........................................... 43
3.1 Swirler nominal values ................................................................................. 49
3.2 Typical performance quantity uncertainty ...................................................... 54
5.1 Manufactured components initial expansion angle .......................................... 65
5.2 Manufactured outer wall parameters ............................................................... 65
5.3 Manufactured tab dimensions ....................................................................... 69
6.1 Diffuser geometry study performance ............................................................. 77
6.2 Swirl study Config. bb performance ............................................................... 87
6.3 Predicted influence of area ratio on recirculation region length with S20 swirl and tabs .... 100
6.4 Adjusted area ratios due to tab blockage ......................................................... 108
6.5 Number of tabs to obtain maximum outlet velocity uniformity ....................... 109
7.1 Turbulence model evaluation summary ........................................................... 111
A.1 Measurement uncertainty sources ................................................................. 134
A.2 Uncertainty contribution by property in axis alignment study for best Config. cc .... 143
B.1 Experiments and CFD strengths and weaknesses ............................................. 147
B.2 Exhaust system design requirements .............................................................. 148
B.3 Optimization study free variable constraints .................................................. 149
B.4 Genetic algorithm grid study performance parameter uncertainty ................. 154
C.1 Summary of CFD executable script files ....................................................... 159
C.2 Config. bb S0 swirl performance using included scripts .................................. 160
E.1 Diffuser geometry study number of retained tests .......................................... 193
E.2 Diffuser geometry study inlet flow blockage .................................................. 194
E.3 Diffuser geometry study experimental performance quantities and standard deviations 198
E.4 Mass flow rate comparison with S0 swirl ....................................................... 200
E.5 Diffuser geometry study diffuser effectiveness .......................... 203
E.6 Diffuser geometry study static pressure recovery breakdown .......................... 204
E.7 Transition duct performance without swirl .......................... 211
E.8 Change in predicted performance due to fully developed turbulent quantity inlet inputs .......................... 211
E.9 Change in predicted performance with S0 swirl due to filleted corner D .......................... 215
E.10 Diffuser geometry study onset of centre-body separation and reattachment points .......................... 217
E.11 Diffuser geometry study static pressure recovery breakdown .......................... 219
E.12 Diffuser geometry study comparison of computational to experimental results .......................... 222
F.1 Swirl study number of retained tests .......................... 223
F.2 Swirl study onset of centre-body separation .......................... 227
F.3 Swirl study outlet velocity uniformity for CBb .......................... 231
F.4 Fine grid study Config. bb performance parameter uncertainty with S20 swirl .......................... 237
F.5 Swirl study Config. bb CTRZ separation and reattachment points .......................... 241
F.6 Swirl study Configs. bb and nb variations in predicted performance with respect to experiment .......................... 242
G.1 Config. bd performance with S0 swirl .......................... 248
G.2 Config. bd performance with S20 swirl .......................... 249
H.1 Config. bb tab study number of retained tests .......................... 264
H.2 Impact of tab orientation on measured inlet quantities .......................... 265
H.3 All OWs tab study number of retained tests .......................... 269
H.4 All OWs tab study with S0 swirl tuft data .......................... 271
H.5 All OWs tab study with S20 swirl onset of centre-body separation .......................... 275
List of Figures

1.1 Annular diffuser cutaway ........................................ 4
2.1 Typical diffuser schematic ........................................ 6
2.2 Generalized annular diffuser map for low inlet blockage .... 7
2.3 Typical jet flow with swirl ....................................... 16
2.4 Effect of pressure gradient on boundary layer profiles ......... 18
2.5 Line of first appreciable stall for annular diffusers .......... 19
2.6 Turbofan exhaust nozzle with lobed mixer schematic ......... 29
2.7 The main flow structures generated by a trapezoidal vortex generator ........................................ 31
2.8 Schematic of a counter-rotating arrangement of vane-type vortex generators in a pipe ................. 37
3.1 Wind tunnel schematic ........................................... 45
3.2 Experimental apparatus ........................................... 46
3.3 Annulus outlet total pressure contours ......................... 47
3.4 Test section schematic depicting instrumentation locations .................. 48
3.5 Traversed swirl profiles ......................................... 48
3.6 Upstream view of instrument locations ......................... 50
3.7 Wall pressure tap orientation .................................... 50
4.1 Symmetry plane computational domain and boundary conditions .................. 56
4.2 Turbulence intensity sensitivity .................................. 59
4.3 2D coarse grid mesh ............................................ 60
4.4 3D coarse 90° wedge outlet mesh ................................ 61
5.1 Manufactured components geometry ............................. 66
5.2 Picture of centre-body and outer wall parts .................... 66
5.3 Area distributions and pressure coefficients for selected configurations ............................. 68
5.4 Square tab schematic ........................................... 69
<table>
<thead>
<tr>
<th>Section</th>
<th>Title</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>6.27</td>
<td>Config. bc with 4 tabs and S20 swirl outlet contours</td>
<td>106</td>
</tr>
<tr>
<td>6.28</td>
<td>Config. bd with 4 tabs and S20 swirl outlet contours</td>
<td>107</td>
</tr>
<tr>
<td>6.29</td>
<td>Tabs study performance with S20 swirl</td>
<td>108</td>
</tr>
<tr>
<td>A.1</td>
<td>Annulus outlet axial velocity profiles</td>
<td>126</td>
</tr>
<tr>
<td>A.2</td>
<td>Annulus outlet tangential velocity profiles</td>
<td>126</td>
</tr>
<tr>
<td>A.3</td>
<td>Annular diffuser inlet axial velocity profiles</td>
<td>127</td>
</tr>
<tr>
<td>A.4</td>
<td>Annular diffuser inlet axial tangential profiles</td>
<td>127</td>
</tr>
<tr>
<td>A.5</td>
<td>3-hole probe calibration curves</td>
<td>128</td>
</tr>
<tr>
<td>A.6</td>
<td>Blower startup pressure transducer and thermocouple moving averages response</td>
<td>131</td>
</tr>
<tr>
<td>A.7</td>
<td>Pressure transducer response to atmospheric conditions</td>
<td>131</td>
</tr>
<tr>
<td>A.8</td>
<td>Pressure transducer response at different sampling frequencies</td>
<td>132</td>
</tr>
<tr>
<td>A.9</td>
<td>Pressure transducer amplitude spectrum</td>
<td>132</td>
</tr>
<tr>
<td>A.10</td>
<td>Pressure transducer response to turbulent flow</td>
<td>133</td>
</tr>
<tr>
<td>A.11</td>
<td>No swirl outlet auxiliary experiment secondary flow vectors</td>
<td>137</td>
</tr>
<tr>
<td>A.12</td>
<td>Total pressure profiles at the annular diffuser inlet</td>
<td>139</td>
</tr>
<tr>
<td>A.13</td>
<td>Axis alignment study outlet contours</td>
<td>141</td>
</tr>
<tr>
<td>A.14</td>
<td>Axis alignment study wall pressure distributions</td>
<td>142</td>
</tr>
<tr>
<td>A.15</td>
<td>Axis shift study outlet contours</td>
<td>144</td>
</tr>
<tr>
<td>A.16</td>
<td>Influence of 3-hole probes in flow on the outlet profile</td>
<td>145</td>
</tr>
<tr>
<td>B.1</td>
<td>Annular diffuser schematic for optimization analysis</td>
<td>149</td>
</tr>
<tr>
<td>B.2</td>
<td>Optimization process schematic</td>
<td>151</td>
</tr>
<tr>
<td>B.3</td>
<td>Optimization study objective space showing all solutions</td>
<td>156</td>
</tr>
<tr>
<td>B.4</td>
<td>Optimization study objective space</td>
<td>157</td>
</tr>
<tr>
<td>B.5</td>
<td>Non-dominated solution geometry profiles after the 35th generation</td>
<td>158</td>
</tr>
<tr>
<td>C.1</td>
<td>Config. bb coarse grid generated from included script</td>
<td>164</td>
</tr>
<tr>
<td>D.1</td>
<td>Transition duct assembly drawing</td>
<td>181</td>
</tr>
<tr>
<td>D.2</td>
<td>Diffuser assembly drawing</td>
<td>181</td>
</tr>
<tr>
<td>E.1</td>
<td>Centre-bodies evaluated by Eckert et al.</td>
<td>192</td>
</tr>
<tr>
<td>E.2</td>
<td>Diffuser geometry study inlet pressure profiles</td>
<td>194</td>
</tr>
<tr>
<td>E.3</td>
<td>Influence of mass flow rate on configurations with S0 swirl static wall pressure distributions</td>
<td>196</td>
</tr>
</tbody>
</table>
E.4 Diffuser geometry study outlet total pressure contours and velocity vectors . . . . . 197
E.5 Influence of Reynolds number on experimental performance in configurations with
S0 swirl performance . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . 199
E.6 Diffuser geometry study objective space . . . . . . . . . . . . . . . . . . . . . . 201
E.7 Influence of initial expansion angle on back pressure coefficient for configurations
with S0 swirl . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . 202
E.8 Typical CFD residual history . . . . . . . . . . . . . . . . . . . . . . . . . . . . . 206
E.9 Typical CFD monitored properties history . . . . . . . . . . . . . . . . . . . . . . 207
E.10 Coarse grid study Config. bb S0 profiles . . . . . . . . . . . . . . . . . . . . . . 208
E.11 Coarse grid study Config. bb S0 wall distributions . . . . . . . . . . . . . . . . . 209
E.12 Influence of coarse grid density on Config. bb performance with S0 swirl . . . . 209
E.13 Influence of Reynolds number on Config. bb performance with S0 swirl . . . . 210
E.14 Input turbulence quantity study predicted Config. bb profiles with S0 swirl . . . . 212
E.15 Input turbulence quantity study predicted influence on Config bb wall pressure dis-
tributions with S0 swirl . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . 213
E.16 Fillet study predicted Config. bb profiles with S0 swirl . . . . . . . . . . . . . . . 214
E.17 Fillet study predicted Config. bb axial shear stress distributions with S0 swirl . . 215
E.18 Influence of turbulence model on inlet pressure profiles with S0 swirl . . . . . . . 216
E.19 Outlet axial velocity profiles with S0 swirl . . . . . . . . . . . . . . . . . . . . . . 218
E.20 Diffuser geometry study objective space . . . . . . . . . . . . . . . . . . . . . . . 220
F.1 Swirl study inlet pressure profiles . . . . . . . . . . . . . . . . . . . . . . . . . . . 224
F.2 Swirl study experimental wall pressure distributions with different CBs . . . . . . 226
F.3 Swirl study outlet axial velocity profiles . . . . . . . . . . . . . . . . . . . . . . . 228
F.4 Swirl study outlet tangential velocity profiles . . . . . . . . . . . . . . . . . . . . . 229
F.5 Swirl study Config. bb. outlet total pressure contours and velocity vectors . . . . . 229
F.6 Swirl study pressure recovery vs. swirl number . . . . . . . . . . . . . . . . . . . . 231
F.7 Swirl study back pressure coefficient vs. initial expansion angle . . . . . . . . . 231
F.8 Swirl study outlet centreline axial velocity vs. initial expansion angle . . . . . . 232
F.9 Swirl study total pressure coefficient vs. initial expansion angle . . . . . . . . . 232
F.10 Swirl study outlet axial vorticity contours and in-plane streamlines for Config. bb . 233
F.11 Swirl study predicted Config. bb profiles with S20 swirl . . . . . . . . . . . . . . 235
F.12 Swirl study predicted Config. bb profiles with S40 swirl . . . . . . . . . . . . . . 236
<table>
<thead>
<tr>
<th>Section</th>
<th>Title</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>F.13</td>
<td>Fine grid study Config. bb S20 SST profiles</td>
<td>238</td>
</tr>
<tr>
<td>F.14</td>
<td>Fine grid study Config. bb S20 SST wall distributions</td>
<td>239</td>
</tr>
<tr>
<td>F.15</td>
<td>Influence of fine grid density on Config. bb performance with S20 swirl</td>
<td>239</td>
</tr>
<tr>
<td>F.16</td>
<td>CFD Swirl study Config. bb inlet axial velocity profiles</td>
<td>240</td>
</tr>
<tr>
<td>F.17</td>
<td>Swirl study Config. bb inlet swirl angle profiles</td>
<td>240</td>
</tr>
<tr>
<td>F.18</td>
<td>Swirl study Configs. bb and nb back pressure vs. swirl number</td>
<td>241</td>
</tr>
<tr>
<td>F.19</td>
<td>Swirl study Configs. bb and nb outlet velocity uniformity vs. swirl number</td>
<td>241</td>
</tr>
<tr>
<td>F.20</td>
<td>Swirl study Configs. bb and nb total pressure loss vs. swirl number</td>
<td>242</td>
</tr>
<tr>
<td>F.21</td>
<td>Influence of swirl on pressure recovery</td>
<td>242</td>
</tr>
<tr>
<td>G.1</td>
<td>Config. bd with S0 swirl experimental outlet total pressure contours and velocity vectors</td>
<td>245</td>
</tr>
<tr>
<td>G.2</td>
<td>Config. bd with S0 swirl outside flow regime flow visualization</td>
<td>246</td>
</tr>
<tr>
<td>G.3</td>
<td>Config. bd with S0 swirl bottom flow regime flow visualization</td>
<td>246</td>
</tr>
<tr>
<td>G.4</td>
<td>Config. bd with S0 swirl experimental wall pressure distributions</td>
<td>247</td>
</tr>
<tr>
<td>G.5</td>
<td>Blower ramp up and down influence on Config. bd</td>
<td>247</td>
</tr>
<tr>
<td>G.6</td>
<td>Two flow modes in Config. bd</td>
<td>247</td>
</tr>
<tr>
<td>G.7</td>
<td>Config. bd wall pressure distributions with S0 swirl</td>
<td>248</td>
</tr>
<tr>
<td>G.8</td>
<td>Config. bd axial velocity profiles with S0 swirl</td>
<td>248</td>
</tr>
<tr>
<td>G.9</td>
<td>Config. bd wall pressure distributions with S20 swirl</td>
<td>249</td>
</tr>
<tr>
<td>G.10</td>
<td>Config. bd outlet axial velocity profiles with S20 swirl</td>
<td>249</td>
</tr>
<tr>
<td>H.1</td>
<td>Configuration bc with 4 tabs and S0 swirl predicted pressure planes and streamlines</td>
<td>251</td>
</tr>
<tr>
<td>H.2</td>
<td>Config. bc with 4 tabs and S0 swirl axial velocity contours and velocity vectors</td>
<td>252</td>
</tr>
<tr>
<td>H.3</td>
<td>Config. bc with 4 tabs and S0 swirl predicted total pressure contours</td>
<td>252</td>
</tr>
<tr>
<td>H.4</td>
<td>Config. bc with 4 tabs and S0 swirl axial vorticity contours and streamlines</td>
<td>253</td>
</tr>
<tr>
<td>H.5</td>
<td>Config. bc with 4 tabs and S0 swirl predicted azimuthal vorticity contours</td>
<td>253</td>
</tr>
<tr>
<td>H.6</td>
<td>Config. bc with 4 tabs and S0 swirl predicted turbulence intensity contours</td>
<td>254</td>
</tr>
<tr>
<td>H.7</td>
<td>CFD history with tabs for a simulation with S20 swirl</td>
<td>256</td>
</tr>
<tr>
<td>H.8</td>
<td>Config. bc with 4 tabs and S0 swirl quadric surface pressure and streamlines</td>
<td>257</td>
</tr>
<tr>
<td>H.9</td>
<td>Config. bd with 4 tabs and S0 swirl quadric surface pressure and streamlines</td>
<td>257</td>
</tr>
<tr>
<td>H.10</td>
<td>Config. bc with 4 tabs and S0 swirl quadric surface axial vorticity</td>
<td>258</td>
</tr>
<tr>
<td>H.11</td>
<td>Inlet pressure profiles for configurations with 4 tabs and S0 swirl</td>
<td>259</td>
</tr>
<tr>
<td>H.12</td>
<td>Wall pressure distributions for configurations with 4 tabs and S0 swirl</td>
<td>260</td>
</tr>
<tr>
<td>------</td>
<td>---------------------------------------------------------------</td>
<td>-----</td>
</tr>
<tr>
<td>H.13</td>
<td>Config. bc with 4 tabs and S0 swirl outlet contours</td>
<td>261</td>
</tr>
<tr>
<td>H.14</td>
<td>Config. bd with 4 tabs and S0 swirl outlet contours</td>
<td>262</td>
</tr>
<tr>
<td>H.15</td>
<td>Tab study performance with S0 swirl</td>
<td>263</td>
</tr>
<tr>
<td>H.16</td>
<td>Tab orientation influence on wall pressure distributions with 4 tabs and S20 swirl</td>
<td>265</td>
</tr>
<tr>
<td>H.17</td>
<td>Tab orientation influence on inlet pressure profiles with 4 tabs and S20 swirl</td>
<td>266</td>
</tr>
<tr>
<td>H.18</td>
<td>Config. bb tab performance with S20 swirl</td>
<td>267</td>
</tr>
<tr>
<td>H.19</td>
<td>Config. bb outlet axial vorticity contours and in-plane streamlines with X tabs and S20 swirl</td>
<td>268</td>
</tr>
<tr>
<td>H.20</td>
<td>All OWs tab study with S0 swirl wall pressure distributions</td>
<td>270</td>
</tr>
<tr>
<td>H.21</td>
<td>All OWs tab study Config. bd outlet total pressure contours and velocity vectors with S0 swirl and 4 tabs</td>
<td>272</td>
</tr>
<tr>
<td>H.22</td>
<td>All OWs tab study with S0 swirl performance</td>
<td>273</td>
</tr>
<tr>
<td>H.23</td>
<td>All OWs tab study with S20 swirl inlet pressure profiles</td>
<td>274</td>
</tr>
<tr>
<td>H.24</td>
<td>All OWs tab study with S20 swirl wall pressure distributions</td>
<td>275</td>
</tr>
<tr>
<td>H.25</td>
<td>All OWs tab study with S20 swirl outlet axial velocity profiles</td>
<td>276</td>
</tr>
<tr>
<td>H.26</td>
<td>All OWs tab study with S20 swirl pressure recovery vs. tab blockage</td>
<td>277</td>
</tr>
<tr>
<td>H.27</td>
<td>All OWs tab study with S20 swirl pressure coefficients vs. area ratio</td>
<td>277</td>
</tr>
</tbody>
</table>
### Nomenclature

- **A** cross section area
- **$A_B$** flow area blockage ratio, $1 - \bar{u}/u_{max}$
- **$A_{tab}$** tab projected blockage area ratio
- **AR** area ratio, $A_e/A_t$
- **$B$** bias
- **$C_1$** forced vortex flow constant
- **$C_2$** free vortex flow constant
- **$C_b$** back pressure coefficient, $(p_{atm} - \bar{p}_t)/\langle q_t \rangle$
- **$C_p$** pressure recovery coefficient, $(\langle p_e \rangle - \langle p_t \rangle)/\langle q_t \rangle$
- **$C_{p,ce}$** conical expansion pressure coefficient, $(\langle p_e \rangle - \langle p_i \rangle)/\langle q_i \rangle$
- **$C_{p,sd}$** solid diffuser pressure coefficient, $(\langle p_e \rangle - \bar{p}_T)/\langle q_T \rangle$
- **$C_p0$** total pressure loss coefficient, $(\langle p_{0,e} \rangle - \langle p_{0,t} \rangle)/\langle q_t \rangle$
- **$C_q$** dynamic pressure coefficient, $(\langle q_e \rangle - \langle q_t \rangle)/\langle q_t \rangle$
- **$C_{\xi_u}$** azimuthal vorticity coefficient, $D_u/\langle u_t \rangle \xi_u$
- **$C_{\xi_x}$** axial vorticity coefficient, $D_u/\langle u_t \rangle \xi_x$
- **CB** centre-body, see Fig. 5.1
- **CFD** computational fluid dynamics
- **CI** confidence interval
- **Config.** Configuration (CB|OW)
- **CTRZ** central toroidal recirculation zone
- **CVP** counter rotating vortex pair
- **$D_h$** hydraulic diameter
- **$D_o$** outer annulus diameter, $\equiv 2R_o$, m
- **FS** full scale
- **GCI** grid convergence index
- **$L$** diffuser length, m
- **$M$** Mach number
- **$M$** number of sets (App. A)
- **MW** molar mass, kg/kmol
- **$N$** number of repetitions per set
- **NEWS** north, east, west, south (looking into flow), see Fig. 3.7
- **$\mathcal{R}$** outer wall, see Fig. 5.1
- **$R_a$** air gas constant
- **$R_i$** inner inlet radius, $\equiv 0.5D_i$
- **$R_o$** outer inlet radius, $\equiv 0.5D_o$
- **RANS** Reynolds-averaged Navier-Stokes
- **$Re$** hydraulic diameter Reynolds number
- **$Re_x$** flat-plate Reynolds number
- **$Rk-\varepsilon$** realizable $k-\varepsilon$
- **RNG** Renormalization group
- **RSS** root-sum-square
- **$S$** swirl number
- **$S$** precision (App. A)
S# swirler, see Tab. 3.1
SST shear stress transport
T static temperature, K
T₀ total temperature, K
TE trailing edge
Tu turbulence intensity
U uncertainty
V velocity magnitude, m/s
VG vortex generator
a speed of sound
a oval semi-major axis length
b oval semi-minor axis length
dx average axial cell length, m
dy₁ wall cell height, m
e eccentricity
h tab height
k turbulence kinetic energy
k specific heat ratio
l turbulence length scale
m mass flow rate, kg/s
p static pressure, Pa(g)
p₀ total pressure, Pa(g)
q dynamic pressure, p₀ − p, Pa
rᵢ radial coordinate at point j
s span
std standard k-ε
u, v, w Cartesian velocity components, m/s
vᵣ radial velocity, m/s
v₀ tangential (swirl) velocity, m/s
w₁ tab width
w_j left (j = l) or right (j = r) sector weight applied to S0 swirl 3-hole probe data
x_j axial coordinate at point j
y⁺ boundary layer dimensionless wall distance

Greek Symbols
α pitch
α₁ first (j = 1), 2nd (j = 2) flow angle above centre-body
αjk angle of panel j–k
β yaw
γ outlet velocity uniformity coefficient, uₑ/(⟨uₑ⟩)
ΔRs annuls height
ΔRᵢ diffuser inlet height, R₀ − Rᵢ
δ boundary layer thickness
δ⁺ displacement thickness
e turbulence dissipation rate
η diffuser effectiveness, C_b/(1 − AR⁻²)
η** locus of maximum diffuser effectiveness at prescribed area ratio
θ cylindrical coordinate angle
φ swirl angle
2θ_d conical diffuser divergence angle
µ dynamic viscosity of air
ξₐ azimuthal vorticity, √(ξ₂⁻ + ξ₄⁻)
ξₚ axial vorticity, ∂w/∂y − ∂v/∂z
ρ density of air
τₗ wall shear stress
φ tab orientation angle
ψ stream function
⟨ ⟩ area-average
⟨ ⟨ ⟩ ⟩ mass flow-weighted average
-c chosen coarse grid
-f chosen fine grid
Subscripts

atm atmosphere

c coarse

cl centreline

e diffuser outlet

exp experiment

f fine

lit literature

max maximum

ref reference

s annulus outlet, \( x = -1.05D_o \)

t annular diffuser inlet, \( x = -0.23D_o \)

Up annulus inlet
Chapter 1

Introduction

Gas turbine exhaust systems are simple and effective devices that are capable of improving performance: for example, reducing the back pressure to increase work generated by a turbomachine or providing more preferable inlet conditions to an afterburner. The design and development of annular diffusers utilized in exhaust systems garnered much attention, particularly since Sovran and Klomp [1] published detailed charts from experimental data of straight-walled conical annular diffusers.

For aerospace applications, it was essential that the exhaust system had minimal weight and fit within the available space. Design difficulties were prevalent for short annular diffusers given that an 8° divergence angle was an upper limit that yielded maximum pressure recovery in unswirled straight-walled annular diffusers [2,3]. Stall did not occur in diffusers with about 10° divergence, constant inner-wall diameter, and length to inlet height ratio less than 6 [4]. Alternatively, Sovran and Klomp [1] reported similar pressure recoveries from a diffuser with 14° divergence that had a negative 10° inner-wall angle. Japikse and Baines [5] acknowledged that pressure recovery coefficients of 0.2–0.5 were tolerated for short diffusers since better performance could not be achieved.

Achieving maximum area ratio in a compact design with a reduced weight was possible by providing a fully open outer diameter exit flow. As an alternative to negative inner-wall angles,
terminating an inner cylinder with a centre-body achieved this design requirement. Previous centre-body studies focused on length and end-shape [6-9]. Wood and Higginbotham [7] suggested that differences in centre-body shape were of secondary importance to that of length; however, Adkins, Jacobsen and Chevalier [9] acknowledged that many centre-bodies were longer than necessary. The added centre-body weight must be supported by larger struts, which impact the exhaust flow and cost to the engine. Optimization tools provided the opportunity to pursue performance benefits from secondary influences, which can offer a competitive advantage.

Breaking from tradition, annular diffusers with initially converging outer wall angles were studied. The application of Thayer [10], Jirásek [11], and Wendt’s [12] curved wall designs was infrared suppression, particularly to eliminate line-of-sight of the turbine. Similarly, Birk and Davis [13] proposed a curved exhaust passage with the objective of reducing or eliminating high radiance sources. Bosioc et al. [14] were interested in mitigating the vortex rope of flow emerging from a Francis turbine runner and consequently investigated a design with a nozzle upstream of a conical diffuser with water jet injection through a centre-body in the nozzle. Numerous experiments were completed on annular diffusers with wall curvature [15-18]. The degree of difficulty in determining the flow distribution was increased because of the radial pressure gradients relating to the geometry.

Better performance in short annular diffusers was possible through implementing augmentation devices, such as vortex generators, on devices with geometrical constraints preventing more preferable flow behaviour or retrofitting poorly designed existing devices. In principle, such flow control devices reduce or eliminate flow separation, thus converting a greater amount of available dynamic pressure into static pressure.

The desire to improve performance of an often poorly designed short annular diffuser that was a component of a turboshaft gas turbine exhaust system intended for aerospace applications was the motivation for this work: specifically, to determine the suitability of implementing passive augmentation devices.
1.1 Objective

The objective was to establish guidelines for three parameters—diffuser geometry, swirl flow angle, and tab placement—that affect pressure recovery and outlet flow distortion in short annular to fully open diffusing passages.

Traditionally, augmentation devices were implemented to prevent flow separation in wide-angled diffusers; therefore, a hypothesis was proposed: vortex generators are beneficial for recovering the energy in a central toroidal recirculation zone caused by a strongly swirling inlet flow.

1.2 Contributions

The focus of this research was on a class of annular diffusers shown in Fig. 1.1 that consisted of conical flow expansion between the centre-body and outer wall and fully open outer diameter (solid) diffusion sections. The contributions of this work were useful for the development of future gas turbine exhaust systems. Specifically, the work:

- Characterized and established guidelines for designing short annular diffusers.
- Developed and integrated numerical optimization methodologies into a short annular diffuser design that had a fully open outer diameter exit flow.
- Assessed the performance of selected annular diffuser configurations experimentally and computationally.
- Evaluated the effect of inlet swirl on performance.
- Determined how to integrate augmentation devices—specifically vortex generating tabs—for improved performance.
The applied research nature of this project provided useful contributions to the engineering community; the fundamental exploration of tabs in a diffuser system with strongly swirling flow was a scientific contribution.

![Annular diffuser cutaway](image)

**Figure 1.1:** Annular diffuser cutaway

1.3 **Scope**

The present study was limited to short annular diffusers typical of those used in aerospace applications. Three parameters that influence diffuser performance were investigated: diffuser geometry, swirl, and passive tab devices. Experiments were completed on a cold-air subsonic flow wind tunnel in the Gas Turbine Laboratory at Queen’s University. Diffuser inlet Reynolds numbers were limited to a maximum of $4 \times 10^5$. Time-averaged pressure data were collected from 3-hole probes at the diffuser inlet, from a 7-hole probe mounted to a traverse table at the outlet, and from wall pressure taps. Numerical simulations were performed with the commercially available software *ANSYS Fluent*® that solved steady-state Reynolds-averaged Navier-Stokes equations on modest grids.
Chapter 2

Theory and Literature Review

Sections 2.1 and 2.2 introduce the fundamentals necessary to evaluate diffuser performance and understand the flow physics. Section 2.3 covers the application of computational fluid dynamics to diffusers and identifies limitations of the numerical methods. Section 2.4 discusses diffuser augmentation as a means of improving performance through countering the naturally occurring flow development within diffusers.

Diffusers have been widely studied where good information is available relating to the influence of non-dimensional length $L/D_t$, area ratio $AR = A_e/A_t \equiv (D_e/D_t)^2$, divergence angle $2\theta_d$, and inlet aerodynamic blockage $A_{B,t}$. Figure 2.1 identifies these variables on a typical axisymmetric conical diffuser. If the cross section is non-circular, the aspect ratio (depth-to-width ratio) has also been well-reviewed. Smaller contributions have been made with respect to Mach number, Reynolds number, turbulence intensity, and vorticity; however, these properties were only loosely understood at best. The emphasis of future research shifted towards numerical studies where good agreement was obtained by tuning the models (such as providing inlet conditions); however, tuning is not particularly useful for design since flow conditions are not a priori known. Furthermore, models broke down if additional components were included in the system since the coupling effects between adjacent components were usually not properly resolved [19].
2.1 Performance Parameters

By design, subsonic diffusers are geometrically simple devices—duct with increasing area—used for converting dynamic pressure to static pressure \([20, 21]\). Two coefficients are frequently used to quantify the pressure performance. Back pressure coefficient is a useful quantity for upstream components:

\[
C_b = \frac{p_{atm} - p_t}{\langle q_t \rangle} \tag{2.1}
\]

where \(p_t\) and \(q_t = p_{0,t} - p_t\) are the static and dynamic pressures at the annular diffuser inlet and \(p_{atm}\) is the atmospheric pressure. Static pressure recovery coefficient,

\[
C_p = \frac{\langle p_e \rangle - \langle p_t \rangle}{\langle q_t \rangle} \tag{2.2}
\]

is a better measure of the diffuser’s pressure recovery, particularly for the present configuration that developed negative outlet pressure, \(p_e\), due to a swirling flow. Mass-flow weighted averaged pressures were used in Eq. (2.2) since experimental outlet data collected from several configurations were blanked and resulted in higher discrepancies when area-averaged. A static pressure coefficient map created by Sovran and Klomp [1] is shown in Fig. 2.2 that can assist designers in the selection...
2.1. PERFORMANCE PARAMETERS

of $AR$ and $L/\Delta R_t$ for developing annular diffusers with maximum pressure recovery.

![Generalized annular diffuser map for low inlet blockage, from [1]. Specific diffusers may depart from general trends. $C_p^\ast = \text{locus of maximum pressure recovery coefficient at prescribed non-dimensional length}; C_p^{**} = \text{locus of maximum pressure recovery coefficient at prescribed area ratio.}](image)

To confirm that the addition of vortex generating tabs converted a greater amount of kinetic energy to pressure, a dynamic pressure coefficient was also evaluated (more negative is better):

$$C_q = \frac{\langle q_e \rangle - \langle q_t \rangle}{\langle q_t \rangle} \quad (2.3)$$

Knowing the flow distortion level leaving the diffuser is critical if downstream devices such as combustors, heat exchanger cores, or regenerators are used [5]. Outlet velocity uniformity was expressed using

$$\gamma = \frac{u_e}{\langle u_e \rangle} \quad (2.4)$$

and centreline velocity $u_{cl}$ (non-dimensionalized by $\langle u_e \rangle$). A more uniform outlet velocity profile
makes better use of the diffuser’s divergence angle and is beneficial for pressure recovery. Literature \[1,5,22\] has proposed to characterize the velocity distribution by a kinetic energy flux factor that is proportional to Eq. (2.4):

$$\gamma_{lit} = \frac{1}{A} \int \left( \frac{u}{u_i} \right)^3 dA \equiv \gamma^{-3/2}$$ (2.5)

Since diffusers are often one component in a larger system, total pressure coefficient,

$$C_{p_0} = \frac{\langle p_{0,e} \rangle - \langle p_{0,t} \rangle}{\langle q_t \rangle}$$ (2.6)

is a qualitative measure of the aerodynamic performance that is influenced by losses such as boundary layer and core flow phenomena; these flow properties are detrimental for one-dimensional analyses. A better designed diffuser will reduce losses in downstream components. In the design stage, Japikse \[23\] encouraged the usage of the loss coefficient $K = C_{p,i} - C_p$; it is equivalent to Eq. (2.6) when inlet and outlet profiles are similar (i.e. uniform).

Prior to the widespread use of computational fluid dynamics, designers calculated diffuser effectiveness,

$$\eta = \frac{C_b}{C_{p,i}}$$ (2.7)

to estimate the performance of an expected new geometry based on similar existing manufactured configurations \[5\]. Ideal pressure recovery was defined by:

$$C_{p,i} = 1 - \frac{1}{AR^2}$$ (2.8)

Maximizing Eq. (2.7) corresponds to minimizing total pressure loss (maximizing Eq. (2.6)).

Equations (2.1), (2.4), and (2.6) were defined to accommodate the optimization process for which it was more convenient to seek out maximum objective values and served as the primary parameters used in this study to evaluate diffuser performance.
2.1.1 Averaging

Area-averaging of property $X$ was achieved by integrating over all of the cells that sub-divided a cross-section:

$$\bar{X} = \frac{1}{A} \sum X_i dA_i$$  \hspace{1cm} (2.9)

Mass-flow weighted averaging (also referred to as mass-averaged) used:

$$\langle X \rangle = \frac{1}{\dot{m}} \sum X_i \rho u_i dA_i$$  \hspace{1cm} (2.10)

where the mass-flow rate was calculated from:

$$\dot{m} = \int \int \rho u_i r dr d\theta = \sum \rho u_i dA_i$$  \hspace{1cm} (2.11)

for gas density $\rho$.

Experimental outlet data were collected from Cartesian coordinate traverses. The data was first interpolated onto a cylindrical coordinate grid of outlet radius $r_e$ using cubic spline interpolation. Conversion of the vertex data into cell-centred data were averaged using weights determined by the length between a cell corner and its centroid.

2.2 Theory

An annular diffuser is a challenging geometry to analyze since many factors that influence the flow development are also dependent on the installation and operating conditions. Inlet fluid characteristics propagate downstream. The geometry itself can alter the natural flowpath. For exhaust system applications, it was of interest to understand the factors that influence pressure recovery and outlet velocity uniformity.
2.2.1 Turbulence

Turbulence is a phenomenon where rapid, random fluctuations occur in the flow due to viscosity: shear stress is typically quantified to measure the fluid’s flow resistance. Initiating turbulence requires a mechanism or structure such as flow geometry or boundary conditions and can strongly influence the large-scale random motion \[24\]; however, the development and final form is only governed by viscous fluid dynamics \[22\]. For example, round jets achieve mixing by means of small-scale viscous mixing at the molecular level in the shear layer \[25\]. Vortices with secondary flow patterns and adverse pressure gradients causing flow separation are turbulent behaviours that may be beneficial for increasing mixing but there is usually an associated loss in performance such as an increase in static back pressure. Specifically, the radial component of turbulence is beneficial in diffusers since it transfers energy towards the walls, which delays separation and improves the radial velocity profile \[5\].

For diffuser flows such as the one depicted in Fig. 2.1, viscosity is often represented by a Reynolds number that is expressed by a ratio of inertial to viscous forces. Internal flow Reynolds numbers were obtained:

\[
Re = \frac{\rho V D_h}{\mu} \tag{2.12}
\]

where \(D_h = 4A/P\) is the hydraulic diameter of a non-circular duct with perimeter \(P\). At the circular annular diffuser inlet, \(Re_t\) was calculated using \(\langle V_t \rangle\) and \(D_{h,t} = D_o - D_i \equiv 2\Delta R_t\). Diffuser flow in aerospace applications is typically fully turbulent so performance can be classified as Reynolds number independent (or very weakly dependent \[5\]) for \(Re_t > 10^5\) \[22\]. Adkins, Jacobsen, and Chevalier \[9\] obtained data for an annular diffuser with only outer wall divergence that showed constant pressure recovery coefficient for \(Re_t > 6 \times 10^4\). Entrance lengths of about 15 diameters, based on the location of the fully-developed pressure gradient, were measured in constant diameter pipes whose inlet flow was turbulent \[26\][27]. For annuli with \(0.28 < D_i/D_o < 0.75\), Jonsson and Sparrow \[28\] measured entrance lengths of 26–29\(D_h\) based on the pressure gradient being within 2% of its fully-developed value.
2.2. THEORY

Turbulence Intensity

Turbulence intensity is a measure of the turbulent velocity fluctuations that are dependent on upstream flow history:

\[ Tu = \frac{\sqrt{\frac{2}{3}k}}{V_{\text{ref}}} \]  

(2.13)

where \( V_{\text{ref}} \) is a time-averaged mean reference velocity. Turbulence kinetic energy, \( k = \frac{1}{2}\overline{u_i u_i} \), is the mean kinetic energy per unit mass in the fluctuating velocity field. As an inlet condition (note that a gas turbine was not modelled) turbulence intensity defines how well-developed the velocity profile is in terms of quantifying the Reynolds stresses. Low turbulence intensity \( Tu < 1\% \) occurs for external flow across aerodynamic objects and higher turbulence \( Tu > 10\% \) is present inside complex geometries and flow inside rotating machinery. For flow in not-so-complex devices like large pipes, the typical range is \( 1\% < Tu < 5\% \) [29]. For annular diffusers, Hestermann et al. [30] and Klein [31] cited that increasing \( Tu \) to 6–8.5\% increased pressure recovery and reduced the amount of separation in a stalled diffuser.

2.2.2 Compressibility

Flow compressibility was quantified by the diffuser inlet Mach number:

\[ M_i = \frac{V_i}{a} \]  

(2.14)

where speed of sound is \( a = \sqrt{kR_a T} \). Air constants are specific heat ratio, \( k = 1.4 \), and gas constant, \( R_a = 288 \text{ J/kg K} \). Although compressibility effects cannot be neglected for \( M_i > 0.3 \), subsonic diffuser flows are typically classified as Mach number independent since it was suggested that a shock must be present (flow must be \( M_i > 0.8–1.1 \)) before performance began to deteriorate [5]. The present study required subsonic flows; therefore, the influence of this parameter was not studied.
2.2.3 Aerodynamic Blockage

Boundary layer height, $\delta$, can be correlated to the amount of energy contained in the near-wall region. As a boundary layer grows, the amount of momentum in the sublayer decreases and results in a velocity deficit. For annular diffusers, aerodynamic blockage,

$$A_B = \frac{2\delta^*}{\Delta R} \quad (2.15)$$

is an indicator for effective cross section area. If $A_B$ is evaluated at the inlet, its influence can be related to pressure recovery. The relationship implies that thinner boundary layers are better for performance. Assuming turbulent flat plate flow, displacement thickness height, $\delta^*$, can be quantified as [32]:

$$\delta^* = 0.02 \frac{x}{Re^{1/7}} \quad (2.16)$$

however its use is questionable in turbomachinery ducts because uniform inlet diffuser flow is unlikely. It is more conventional to express flow blockage as:

$$A_B = 1 - \frac{\pi}{u_{max}} \quad (2.17)$$

where a more uniform velocity profile correlates to higher pressure recovery.

The influence of aerodynamic blockage was not well understood in annular diffusers. Several experiments [23, 33, 34] reported that pressure recovery initially decreased with increased blockage but increased for long inlet lengths that achieved fully-developed flow. It was suggested that higher order effects such as turbulence intensity and boundary layer mixing may affect results. The challenge of isolating the influence of inlet conditions is even more complicated since the boundary layers on the hub and casing surfaces can have significantly different histories that develop complex interactions. For example, the boundary layer effect on one wall can influence the boundary layer growth on the opposite wall [5].
2.2.4 Swirl

Swirl was especially relevant in this study since the rotation rate of the exhaust gases leaving the aircraft power system vary by operating condition. Swirl angle was calculated from:

$$\vartheta = \tan^{-1} \frac{v_\theta}{u}$$  \hspace{0.5cm} (2.18)

Circulating flows are characterized by two types [35]:

- Free vortex flow. An object travelling on a circular path does not rotate. The flow is irrotational. Particles farther away from the centre take longer to travel around the circumference—

$$rv_\theta = C_2$$  \hspace{0.5cm} where  \hspace{0.5cm} C_2 \hspace{0.5cm} \text{is a constant.}  \hspace{0.5cm} \text{This is analogous to bathtub vortex motion that is produced by flat-vane swirlers.}$$

- Forced vortex flow. Solid (rigid) body rotation whereby an object rotates as it travels on a circular path. Since tangential velocity increases with increasing radius—

$$v_\theta/r = C_1$$  \hspace{0.5cm} where  \hspace{0.5cm} C_1 \hspace{0.5cm} \text{is a constant—all particles at their respective radii complete a revolution in the same time.}$$

This flow is noted for being more aerodynamic and can be produced by twisted-vane swirlers.

It is common for swirling flow to be described as a combination of free- and forced-vortex motion.

Lohmann, Markowski, and Brookman [36] commented that for swirling flows, the influences of inlet swirl angle and wall angle on pressure recovery overwhelmed those of area ratio and nondimensional length. Mechanisms from all three coordinate directions affect static pressure:

- A swirling flow develops a centrifugal force that is balanced by a radial pressure gradient.

From radial equilibrium on an element \(d\theta \times dr\) with \(v_r = 0\):

$$\frac{\partial p}{\partial r} = \frac{\rho v^2_\theta}{r}$$  \hspace{0.5cm} (2.19)
Absence of friction, $v_{θr} = C$: a decrease in tangential velocity momentum in an outward canted diffuser due to an increase in flow area causes static pressure to increase.

- The diffusion of the streamwise velocity component is affected since it is coupled with the tangential flow.

The latter two mechanisms are primarily responsible for determining the pressure recovery coefficient.

Swirl can be generated experimentally in a radial inflow plane and with axial blade cascades. Axial cascades can more closely simulate turbomachinery flow conditions but were often simple fixed flat vanes of poor design—it may be difficult to isolate the influence of swirl from cascades that also develop different turbulence intensity, velocity or total pressure gradients, vorticity or wake shedding, and inlet aerodynamic blockage [5].

Dovzhik and Kartavenko [37] tested flat and twisted vane swirlers with nominal $10^\circ$ incremental inlet swirl angles between $0$ and $40^\circ$ and reported that the most efficient pressure recovery for annular diffusers with $1.5^\circ$ inner wall and $8^\circ$ outer wall angles, and $0^\circ$ inner and $6^\circ$ outer wall angles occurred at $10^\circ$ swirl. Fleige et al.’s [38] diffuser and study were similar and also reported the most pressure recovery with either none or $8^\circ$ swirl. Eckert et al.’s [8] device had a $2.6^\circ$ outer wall angle and elliptical centre-body that gave an equivalent cone angle of $11.5^\circ$ and produced greater pressure recovery for swirl angles between $10^\circ$ and $14^\circ$ than those below $10^\circ$.

Coladipietro et al. [33] reported for diffusers with both inner and outer wall angles of $20^\circ$ that more pressure recovery occurred for larger swirl angles up to $20^\circ$ swirl. Lohmann et al. [35] tested inlet swirl angles of $0$, $30$, and $48^\circ$ on eight different diffuser configurations and found that for swirl angle below $30^\circ$, maximum pressure recovery occurred when the inner wall angle was around $0^\circ$ for $AR = 1.5$ diffusers. Klomp [21] performed a more comprehensive study that considered 38 configurations of varying inner and outer cone angles and length with inlet swirl up to $44^\circ$ where most designs were near an optimum geometry. More swirl reduced the inlet velocity deficit near the outer wall but increased the deficit near the inner wall and separation advanced towards the inlet.
Violent vibrations were created when the inlet profiles were perturbed by enough swirl. An annular diffuser with a negative inner wall angle had the largest negative impact since inner wall separation occurred even at relatively small swirl angles.

Overall, most studies reported acceptable amounts of pressure recovery for swirl angle below 30°, and a rapid loss thereafter. Correlations using experimental data from open literature that covered basic diffuser geometry, inlet swirl, and inlet aerodynamic blockage were developed by Japikse [39].

**Swirl Classification**

The degree of swirl generated by a swirler was quantified by the (alternate) swirl number [35, 40]:

\[
S = \frac{G_\theta}{G_x L}
\]  

(2.20)

The axial flux of angular momentum was expressed as:

\[
G_\theta = \int \rho u v_\theta r^2 dr
\]  

(2.21)

and the axial flux of axial momentum neglecting the pressure term was:

\[
G_x = \int \rho u^2 r dr
\]  

(2.22)

for time-averaged axial and tangential velocities, \( u \) and \( v_\theta \).

For annular flows, there was no clear consensus on the selection of the characteristic length \( L \) in Eq. (2.20). Choices included \( D_h/2 \) [41, 42], \( R_o \) [43–45], \( R_i \) [46], \( (R_o+R_i)/2 \) [47], and \( 2/3 \left( R_o^3-R_i^3 \right) / \left( R_o^3-R_i^3 \right) \) [48].

The characteristic length \( L = D_h/2 \) was implemented since it gave the best agreement to the swirl flow characterizations developed by Gupta [35]:
• very weak  \( S \lesssim 0.2 \) – pressure gradients may be omitted

• weak  \( S \lesssim 0.4 \)  – low degree of swirl but radial pressure gradients \( \frac{\partial p}{\partial r} = \rho v^2 \theta / r \) may be significant in a straight duct
  – nonreacting flow can be treated as isotropic

• strong  \( S \gtrsim 0.6 \)  – higher degrees of swirl with strong radial and axial pressure gradients occurring near the exit produce an axial central toroidal recirculation zone (CTRZ)
  – nonisotropic

Precise swirl number quantities that distinguish the amount of swirl vary by application; the range \( 0.4 < S < 0.6 \) may be characterized as either weak or strong swirl. Figures 2.3(a) and (b) are typical jet flows with respectively low- and high-degrees of swirl.

Figure 2.3: Typical jet flow with swirl, from [35]
2.2.5 Diffusion

Since mass must be conserved in subsonic diffusers, as the flow area increases, velocity decreases. And from conservation of linear momentum, a decrease in velocity is balanced by an increase in static pressure. In real flows, viscous forces are present, so diffuser flow cannot be treated as one-dimensional. For axisymmetric annular diffusers, symmetry can be assumed in numerical analyses; however, flow velocity at the wall is zero. Near walls, the streamwise (i.e., axial) momentum equation reduces to:

\[ \tau_w = \mu \frac{\partial^2 u}{\partial y^2} = \frac{\partial p}{\partial x} \]  

(2.23)

For positive pressure gradient, the second derivative of velocity must be positive at the wall; however, it must be negative at the outer layer (where \( y = \delta \)) in order for the fluid to merge smoothly with the mainstream flow where the streamwise velocity is \( U(x) \). It follows that the second derivative must pass through zero within the boundary layer and subsequently exhibit a characteristic S-shape [32]. Figure 2.4 shows that as flow progresses downstream with an initially favourable pressure gradient, the point of inflection rises towards the surface and enters into the flow. The continuing adverse pressure gradient causes an onset of flow separation, \( \tau_w = 0 \), and eventual flow separation.

An additional consequence of decelerating flow in diverging ducts is thickening boundary layers [5, 22]. Unlike boundary layers in the presence of favourable pressure gradients that can be characterized by a logarithmic velocity profile, increasing departures from the law of the wall with respect to axial location were observed from velocity profiles within a diffuser by Stevens and Fry [15] and Stevens and Williams [34]. Since diffuser boundary layers were not in equilibrium, there were distinct differences in the rates of production and dissipation of kinetic energy. Japikse and Baines [5] suggested that diffuser performance was more strongly correlated to turbulence distribution than other effects; therefore, avoiding the natural boundary layer growth was desirable.

Annular diffuser design is more complex because of the additional design variables. In addition to length and inlet radius ratio, the inner and outer wall angles are required to determine the area
CHAPTER 2. THEORY AND LITERATURE REVIEW

Figure 2.4: Effect of pressure gradient on boundary layer profiles. PI = point of inflection, from White [32].

ratio [1]. For example, Ishikawa and Nakamura [49] found significant difference in performance between outward parallel and diverging walls (different wall angles) for $L/R_o > 2$. Johnston [2] noted the challenges associated with the presence of an adverse pressure gradient and concluded that better performance in conical annular diffusers was achieved with a uniform inlet velocity profile and that performance decreased for divergence angles greater than 15°.

Diffuser Stall Regimes

The diffuser operating flow regime is strongly dependent on divergence angle and length along with the inlet conditions. Figure 2.5 distinguishes the stall regimes for straight-cored annular diffusers. As divergence angle increases for a given length, the initially uninstalled flow first experiences appreciable stall: thickening boundary layers and small regions of separation in the corners. Next, large transitory stall occurs whereby regions of reversed flow oscillate from one side to the other. Fully developed stall results in a stable recirculating region along a wall. Finally, jet flow is characterized by fully separated regions along both walls [22].
Cherry et al. [50] noted that detailed velocity patterns within the diffuser were difficult to acquire; however, they successfully implemented magnetic resonance velocimetry to confirm the consequences of thickening boundary layers. By reducing the diffuser’s effective area ratio, pressure recovery decreased. Adenubi [51] tested annular diffusers with straight inner walls and outer walls angled at $5^\circ$, $10^\circ$, and $15^\circ$ with area ratios of 1.47, 2.0, and 2.6 respectively. Stall patterns were characterized using 2.5 cm wool tufts that were attached to the walls in a random pattern. The smaller diffusers did not stall; however, incipient or intermittent transitory stall occurred in the $15^\circ$ diffuser. Highest pressure recovery occurred in the $10^\circ$ diffuser whereas the $15^\circ$ diffuser obtained a $C_p$ 8% lower than its predicted value.

Maximum pressure recovery is usually achieved with designs operating near separation. For example, Adkins [52] suggested that quasi-stable flow (eddies formed by transitory stall) assisted in the diffusion process by moving more energetic main flow into the boundary layer.

The aforementioned stall definitions apply to two-dimensional diffusers; however, they are also applicable to annular diffusers used in the present study with fully open outer diameter outlets.
Transitory or fully developed stall can occur in the core region and may be further exacerbated due to swirl. In the presence of swirling flow, centrifugal forces push the flow outwards and can result in a CTRZ. Wider geometries can achieve jet-like flow conditions where flow separates from a portion of the outer wall and passes through the core region instead. A hysteresis zone exists between the fully-developed stall and jet-flow regimes that can result in either of the two stable stall regimes for the given geometry.

2.2.6 Curvature

Wall radius of curvature, $R$, has a major effect on the development of a boundary layer. Applying curvature to the diffuser inlet is beneficial to prevent flow separation since flow typically transitions from a constant-radius upstream duct into a wall-angled diffuser. In general, sudden changes in geometry, such as kinks, should be avoided to ensure smooth transitions between components [5]. In instances where the flow must be turned, maximum curvatures should be applied near the inlet where the boundary layer is still relatively thin [5].

The challenge involved with turning flow is the radial pressure gradient that causes increasing static pressure with bend radius and the corresponding decrease in velocity from the inside bend (convex curvature) to the outside bend (concave curvature). When the flow is turned, the presence of centrifugal and pressure forces deflect the faster moving core towards the outer bend. In comparison, slower energy-deficient fluid nearer to the outer bend is incapable of travelling through the developed adverse pressure gradients, but instead moves around the walls towards the low static pressure region on the inside bend; this movement of low-energy fluid, in addition to the deflection of the high-velocity core region, sets up two cells of secondary flow that are normal to the axial direction [20].

Slight curvatures in a mixing section have few unfavourable effects on performance [53] but the $C_p$ of even well-designed bent diffusers is lower than comparable straight-walled diffusers [5]. Gradual bends do not obtain the same benefits because the increase in skin friction is offset by the
decrease in cross sectional area. For sharp bends (radius of curvature to width ratio less than 2 and bend angles greater than $45^\circ$), turning vanes can be inserted to reduce pressure loss and to provide more uniform flow at the bend exit, since the separate channels keep more flow nearer to the inner radius [22].

### 2.2.7 Struts

Given that a horizontal wind tunnel was used in the present study, struts were present to support the inner cylinder of the annulus. If struts are not aligned with the flow direction, they may act as bluff bodies and cause unsteady wakes and acoustic noise. In the absence of swirl, wakes propagate downstream from aligned struts, usually with minimal impact on pressure recovery. Tapering struts (by linearly varying the chord and thickness along the span) was found to reduce vortex shedding amplitude and was beneficial to performance over a swirl angle range from $30^\circ$ to $60^\circ$ [54].

Cylindrical struts are an alternative that do not significantly change the flow structure but increased blockage near the hub displaces more flow towards the casing. Fleige et al. [38] found that symmetrical airfoil struts generated more pressure recovery for inlet swirl angle below $10^\circ$ but cylindrical struts were better for higher swirl angles. Higher recovery always occurred without struts. With struts at low swirl angles, the recirculation bubble length in an annular diffuser was smaller than a corresponding case without struts because of the likely additional production of turbulent kinetic energy in the strut wakes. Fortunately, the present test section did not require additional supports; therefore, evaluating the behaviour of swirling flow around struts was avoided.

### 2.3 Computational Fluid Dynamics

A goal of research and development is to either develop new devices or discover and create new knowledge about scientific and technological topics [55]. Although creativity serves an important
role in the design process, tools are essential to instill confidence in the product development. Computational fluid dynamics (CFD) has become an important component in solving problems involving fluid flow and heat transfer.

In tandem with early experimental efforts that designed geometries based on performance maps, methodologies were developed to create analytical solutions to diffuser flow [3, 23, 52]. It was common to assume an inviscid core flow; however, the growth of the boundary layer cannot be neglected as it is a major factor in determining diffuser performance. Simple models proposed iterative decoupled or weakly coupled processes that traditionally implemented Prandtl’s boundary layer approximation [56]. This method was noted for working well if the interaction between the wall shear layer and external flow was relatively weak, such as up to the point of separation in straight-walled diffusers. Modifications to simple models included different correlations to solve the viscous forces (such as the Ludwig-Tillmann skin friction coefficient [56]), accounting for radial curvature, aspect ratio corrections, and inlet velocity gradients; however, these models only successfully modelled unstalled diffusers.

In the late 1970s, more advanced analytical models were introduced to calculate boundary layer thickness, shape factor, and predict flow separation since the models were free of singularities. Interactive solution techniques were required since the core flow and boundary layer equations were strongly coupled. Both two-dimensional and three-dimensional boundary layer techniques were developed where the third dimension can accommodate axisymmetric swirl [5]. Successful agreement was often achieved between experiments and the numerical models, but the models were not universally valid due to the inherent assumptions and challenges in initializing the more complex models.

Implementing CFD is now more convenient and economical. The successful use of the software is not straightforward since a designer must make choices between various parameters because more accurate representations are also usually more computationally complex. Experience is essential to lead to the best compromise where typical questions necessary to conduct a CFD analysis include:
2.3. COMPUTATIONAL FLUID DYNAMICS

- Can subsonic flows be evaluated with incompressible flow equations?
- Is it necessary to develop a mesh that resolves all of the geometrical features, such as curvature?
- Are secondary flows dominant in an axisymmetric device, or can a 2D geometry be simulated?

Even with a recent rapid increase in affordable computational power, the Reynolds-averaged Navier-Stokes (RANS) equations are generally preferred for application-type problems. For example, RANS equations have gained popularity in aircraft design problems. Solutions not modelling heat transfer are obtained from the set of mass and momentum equations:

\[
\frac{\partial \rho}{\partial t} + \frac{\partial \rho u_j}{\partial x_j} = 0 \tag{2.24}
\]

\[
\frac{\partial \rho u_i}{\partial t} + \frac{\partial \rho u_j u_i}{\partial x_j} = f_i - \frac{\partial p}{\partial x_i} + \frac{\partial \tau_{ij}}{\partial x_j} + \frac{\partial (-\rho u'_i u'_j)}{\partial x_j} \tag{2.25}
\]

for time-averaged velocities \( u_i \). The vector \( f_i \) represents external forces (such as gravity or buoyancy), \( \tau_{ij} \) is the viscous stress tensor, and \( \partial / \partial t = 0 \) for steady flow.

More accurate solutions obtained from large eddy simulation (LES) \[57\] or direct numerical simulation \[58\] do not necessarily add value to the design process and cannot currently be practically obtained. The design process in a time-constrained development program is more concerned with achieving the required level of accuracy and reliability to adequately simulate the relevant flow phenomena. In general, the simpler the flow physics, the more robust and reliable the solution process \[59\]. This means that although CFD can be applied to any flow problem, there are limitations that are compounded due to the requirement of a turbulence model to bring closure to the RANS equations and grid design. Notwithstanding, analytic integral methods should not be ignored because many applications can still benefit from these simpler models, that in some cases, are capable of achieving better accuracy than most Navier-Stokes solutions \[5\].

Many investigations involving CFD with the RANS equations of diffusing flows are available
Results for diffusing flows have been obtained to describe the influence of geometry modifications \[60\text{-}65\], curvature \[66\], turbulence intensity \[67\], swirl \[38\text{-}60,68\text{-}73\], vortex generators \[12,74\text{-}76\], and turbulence model \[62,73,77\]. Typically, justification was given on the selection of turbulence model where either renormalization group (RNG), realizable $k$-$\varepsilon$ ($R_k$-$\varepsilon$), or shear stress transport (SST) were implemented.

The Energy Systems group at Queen’s University has produced numerous similar investigations to the present annular diffuser study \[78\text{-}82\]. It was found that two-equation turbulence models were not as good at predicting flow with swirl angles greater than $20^\circ$ or for diffusers with divergence angles $2\theta_d > 10^\circ$. The numerical solutions over-predicted pressure recovery in the diffusers as a consequence of the isotropic assumption that led to over-predictions in upstream values. Regardless, some successes were reported where the $R_k$-$\varepsilon$ turbulence model was generally regarded as providing the best solution in terms of numerical accuracy and economy.

Often, CFD papers also include experimental data for validation. Moderate agreement was achieved for performance parameters but local flow features were commonly poorly predicted, such as an over-prediction in size of a separation zone—this was particularly prevalent for wider-angled diffusers. Cherry et al. \[64\] obtained reasonable agreement using LES in the prediction and strength of the recirculation in their tested diffusers. Vassiliev et al. \[62\] selected annular diffusers evaluated by Sovran and Klomp \[1\] and Stevens and Williams \[34\] with $1.2 < AR < 5.9$ and conical equivalents $12^\circ < 2\theta_d < 31^\circ$. Better comparison of the $k$-$\varepsilon$ models was achieved with the $R_k$-$\varepsilon$ model and two-zone wall treatment: reasonable predictions were obtained for pressure recovery with respect to area ratio, opening angle, and diffuser length.

### 2.3.1 Turbulence Models

The turbulence models that were used are referred to as statistical turbulence models since the original Navier-Stokes equations were modified through the introduction of time-averaged and fluctuating quantities to produce the RANS equations. Numerous turbulence models were developed
to account for the fluctuating quantities (the Reynolds stresses \(-\rho \overline{u_i' u_j'}\) in Eq. (2.24)). In particular, two-equation turbulence models were popular. The well-established \(k-\varepsilon\) model was proven to be stable and numerically robust for many flow problems where \(k\) is the turbulence kinetic energy:

\[
k = \frac{1}{2} \overline{u_i' u_i'}
\]

and \(\varepsilon\) is turbulence eddy dissipation:

\[
\varepsilon = -\frac{\mu}{\rho} \frac{\partial \overline{u_i' u_j'}}{\partial x_j} \frac{\partial \overline{u_i' u_i'}}{\partial x_i}
\]

The basic models for \(k\) and \(\varepsilon\) were not tuned for flows including curvature, low Reynolds number, and near wall phenomena [83]; therefore, modifications were available [48].

Three \(k-\varepsilon\) turbulence models were available in Fluent [84]: standard, RNG, and realizable. The standard model [85] was noted as being slightly over-diffusive in certain situations whereas the renormalization group (RNG) model [86] was specifically designed to reduce turbulent viscosity in response to high rates of strain and streamline curvature. For swirling flows, solutions calculated with the RNG model were more representative than those calculated with the standard \(k-\varepsilon\) model because the RNG model included a modification dependent on the swirl that was applied to the turbulent viscosity calculation [48].

Development of the R\(k-\varepsilon\) model was similar to the standard model but with two exceptions: (1) the turbulent viscosity formulation,

\[
\mu_t = \rho C_\mu \frac{k^2}{\varepsilon}
\]

was based on a variable \(C_\mu\) that was a function of mean strain and rotation rates, angular velocity, and the turbulence fields; and (2) the dissipation rate transport equation was derived from an exact equation for the transport of mean-square vorticity fluctuation [84].

Realizability means equating positive Reynolds normal stresses in all directions. This is not
guaranteed from the Boussinesq model where comparison of the one-equation to two-equation turbulence viscosity models gives:

\[ \varepsilon = C_{\mu}^{3/4} \mu^{1.5} L \]  

(2.29)

Moore and Moore [87] noted that in the freestream region of turbomachinery flow, typical magnitudes are \( \frac{T u}{D} = 1\% \), \( L = 0.04 D \) (e.g. \( D \), the pitch), and \( C_{\mu} = 0.09 \). An acceleration in the pitch direction, \( \alpha \), of \( \frac{\partial U}{\partial x_{\alpha}} > 0.2 \frac{U_{o}}{D} \) is sufficient to cause negative modelled normal stress. This can lead to large erroneous growth in \( k \) near the leading edge of an airfoil for example.

Both the RNG and R\( k \)-\( \varepsilon \) models have substantial improvements over the standard \( k \)-\( \varepsilon \) model in obtaining better predictions of flows involving strong streamline curvature, vortices, and rotation. Preferential selection between the two has not yet been established [48]; however, the R\( k \)-\( \varepsilon \) turbulence model was well-suited for more accurately predicting the spreading rate of round jets as well as providing better performance for flows involving boundary layers under strong adverse pressure gradients, separation, complex secondary flow features, and recirculation caused by backwards facing steps [88].

Alternatively, Menter’s shear stress transport (SST) model also had some merit for implementation to the present diffusing flow problem. The SST model used \( k \)-\( \omega \) for near wall treatment and \( k \)-\( \varepsilon \) in the outer wake region and in free shear layers [89]. This methodology did not require wall functions but instead implemented blending functions to ensure a smooth transition between the two flow regions. The SST model obtained more representative solutions than the other turbulence models for flows dominated by boundary layer behaviour, including applications where wall heat transfer was important and adverse pressure gradients and flow separation were present [90]. Preference was given to the SST model in several studies: Dudek [91] simulated the effects of vane-type vortex generators in ducts; Babu et al. [63] simulated boundary layer control in turbine exhaust diffusers; and Zhang et al. [73] modelled an inter-turbine transition duct.

The Reynolds Stress Model (RSM) [92] was an anisotropic turbulence model with the potential to characterize a more representative flow-field. Previous efforts in the Energy Systems group had
2.4. DIFFUSER AUGMENTATION

little success obtaining convergence or yielding better solutions with RSM in comparison to results obtained using $k$-$\varepsilon$ models. Moreover, RSM had not been thoroughly tested for recirculating and swirling flows [83]. The RSM model was gaining wider acceptance for swirling diffusing flows but discrepancies such as incorrect circumferential velocity profiles still occurred [72]. Kaci et al. [93] simulated tabs in a straight tube and noted that the RSM model did not offer strong advantages in the high-shear region and so gave preference to the standard $k$-$\varepsilon$ model.

2.3.2 Wall Treatment

Two options were available in Fluent [84] for turbulence models requiring near-wall treatment: (1) coarse grids rely on wall functions to calculate boundary layers and (2) near wall refinement. Of the various wall functions available, non-equilibrium wall functions [94] were generally preferred for diffusing flows. These functions were sensitized to pressure gradient effects and were recommended for use in complex flows involving separation, reattachment, impingement, and swirl [48]. It was preferable when implementing wall functions to locate the first cell at a distance of $y^+ \approx 30–50$.

Enhanced wall treatment was a two-layer near-wall modelling approach that was capable of completely resolving the viscosity-affected near-wall region all the way to the viscous sublayer ($y^+ \approx 1$). Wolfstein’s [95] one-equation model was employed in the near-wall region and the fully turbulent region used the equations provided by the turbulence model. A smooth transition of turbulent viscosity between the two regions used Jongen’s [96] blending function. Acceptable mesh design required the first cell from a wall within $y^+ < 5$ and at least 10 structured cells in the boundary layer; this was also the case when the SST turbulence model was implemented.

2.4 Diffuser Augmentation

Minimizing naturally occurring separation by adverse pressure gradients or swirl by passive means in simple diffusers can be achieved using diffuser augmentation. Several techniques were previously
explored to re-energize the boundary layer: vortex generators, wall suction, blowing, increasing wall surface roughness, screens, and vanes or splitters. In most cases, the device augmented the level of turbulent mixing, particularly in stabilizing the boundary layer. The limitation to using augmentation devices is that the diffuser must have a wide angle and experience stall; otherwise the devices will either be of no value or detrimental to the diffuser pressure recovery [5] since including tabs, for example, necessarily increases total pressure loss.

It was not clear from literature if pressure can be recovered in a less aggressive diffuser with vortex generators by reducing the CTRZ created by a swirling inlet flow. Augmentation devices that were evaluated in the presence of swirl typically involved aircraft S-shaped air intake ducts that reported engine surge at high angles of attack. Seddon [97] positioned fences on the first bend and discovered a configuration that reduced swirl and outlet distortion. Fundamentally, this was a different problem since swirl (more appropriately defined as secondary flow since it was self-generated and was exacerbated at high angles of attack) occurred due to the action of a transverse pressure gradient set up in the first bend of the duct.

Investigations involving augmentation devices in the presence of swirling flow was limited. Wright et al. [98] was a noteworthy exception with several additional investigations (Lei et al. [99, 100]) that evaluated the effect of swirl in a turbofan engine exhaust nozzle with a lobed mixer that is schematically shown in Fig. 2.6. The purpose of the lobes was to improve mixing from the production of strong vortices but also acted to remove swirl in the core flow. CFD complemented experimental results that reported improved performance with 10° swirl but improved mixing was at the expense of higher pressure and thrust losses with 30° swirl.

Augmentation for any diffuser is merited if the purpose is to reduce outlet flow distortion. Applications that can benefit from more uniform velocity profiles leaving the diffuser include aircraft engine intake ducts [97, 101, 102], interstage ducts between gas generator turbines and power turbines [16, 18, 73] and exhaust systems designed for infrared suppression [10, 11, 103]. Carletti, Rogers, and Parekh [25] noted that their vortex generators encouraged centreline velocity decay by
2.4. DIFFUSER AUGMENTATION

2.4.1 Vortex Generators

Vortex generators (VG) are passive mixing obstructions placed in the flow path that can be conveniently retrofit to existing devices. Some VGs produce a single vortex whereas tabs (a type of VG) produce a counter-rotating pair of vortices. The design principle of the generators is to force fluid particles to travel in the streamwise direction along helical paths and pull high momentum fluid from the core flow toward the boundary surface. In doing so, the boundary layer is re-energized, which reduces flow distortion and delays flow separation that would otherwise occur due to the natural boundary layer growth caused by surface friction. Since the strongest effect is usually just downstream of the obstruction, VGs are usually placed just upstream of the region with the strongest adverse pressure gradient [5].
The specific contributions of the vortical structures generated by VG tabs to the mixing process was investigated by Habchi et al. [104]. Trapezoidal tabs were placed in a circular pipe and instantaneous velocities were acquired with laser doppler velocimetry. The instrument was only able to measure the axial Reynolds stresses; however, CFD was also conducted with the RSM turbulence model to complete the description. Evidence from literature along with the measured results validated the simulations.

Figure 2.7 is a schematic showing the main flow structures generated by a tab. (Note that the discussion associated with the figure identifies $z$ as the axial direction.) The necklace vortex observed upstream of the tab develops due to the interaction between the main flow and low-momentum fluid in the corner between the tab and pipe wall. High-velocity gradients are responsible for generating transversal vorticity. In the near wake of the tab, a dead zone develops with transverse vortices since the flow stream must detour around the obstacle. The longitudinal counter-rotating vortex pair (CVP) forms in the wake of the tab due to the pressure difference between the low-momentum region under the tab and high-momentum region above. A saddle point of separation occurs on the tab symmetry plane and stable foci are present in each CVP. The CVP is primarily responsible for macro-mixing whose strength exponentially decays with downstream distance. Defining a vanished vortex when its circulation has reached 50% of its initial value, both experiment and CFD determined that the CVP vortices vanished at $z \approx 5h$ ($z=$axial distance, $h=$tab height).

Hairpin (horseshoe) vortices are transient structures that ride on top of the CVP and contribute to the meso-mixing process. The hairpin legs interact with the near-wall region and as the hairpin heads are convected downstream, they move away from the wall and eventually vanish around $z \approx 8h$. The reverse vortices have lower strength and vorticity opposite to the hairpins and are damped at $z \approx 3h$.

Kaci et al. [93] equipped a cylindrical tube with seven rows of streamwise tabs and observed that the longitudinal vortices (or CVP) are the major mixing mechanism. Turbulence kinetic energy dissipation rate was increased by factors up to 40 over simple turbulent pipe flow.
Quantifying Vorticity

Dominant structures identified in Fig. 2.7 include counter-rotating and hairpin vortices. Identifying these numerically was possible using axial vorticity for the CVP:

\[ \xi_x = \left( \frac{\partial w}{\partial y} - \frac{\partial v}{\partial z} \right) \]  

(2.30)

and azimuthal vorticity for the hairpins:

\[ \xi_a = \sqrt{\xi_{\theta}^2 + \xi_r^2} \]  

(2.31)
In Cartesian coordinates, tangential vorticity is:

$$\xi_\theta = \frac{1}{r} \left[ \left( y \frac{\partial v}{\partial x} + z \frac{\partial w}{\partial x} \right) - \left( y \frac{\partial u}{\partial y} + z \frac{\partial u}{\partial z} \right) \right]$$

and radial vorticity is:

$$\xi_r = \frac{1}{r} \left[ \left( -z \frac{\partial u}{\partial y} + y \frac{\partial u}{\partial z} \right) - \left( y \frac{\partial w}{\partial x} - z \frac{\partial v}{\partial x} \right) \right]$$

for $r = \sqrt{y^2 + z^2}$.

Gregory-Smith, Graves, and Walsh [105] recognized that the incompressible Helmoltz’ equation could be used to calculate expressions for the tangential and radial components of vorticity from a single plane of experimental outlet data:

$$\xi_\theta = \frac{1}{u} \left( v \xi_x - \frac{1}{r} \frac{\partial p_0}{\partial r} \right)$$

$$\xi_r = \frac{1}{u} \left( v \xi_x + \frac{1}{\rho r} \frac{\partial p_0}{\partial \theta} \right)$$

This is an alternative to completing multiple traverses where it can be challenging to obtain reasonable axial spacing for accurate representation of the $\partial / \partial x$ gradients.

The general coefficient implemented to normalize axial vorticity was:

$$C_{\xi_x} = \frac{D_{ht}}{\langle u_t \rangle} \xi_x$$

but for outlet axial vorticity:

$$C_{\xi_x,e} = \frac{r_e}{\langle V_e \rangle} \xi_x$$

Azimuthal vorticity coefficient was defined by:

$$C_{\xi_a} = \frac{D_{ht}}{\langle u_t \rangle} \xi_a$$

Circulation is an indication of overall vorticity strength. Assuming symmetry about a tab, the
2.4. DIFFUSER AUGMENTATION

strength of the CW structure (looking upstream) can be found by integrating on the right half of the normal cross-section and the CCW structure strength can be calculated using the left half. The problem with this for flow through a duct is that the boundary layers, which in some places have higher vorticity, are included in the integration. Instead, strength in the present investigation was determined from maximum vertex values. The $\xi_x$ strength was determined from a parallel plane 5 mm above the centre-body and $\xi_\theta$ along a $\theta$-plane passing through the centre of the tab.

Nozzle Applications

The advent of placing tabs at nozzle outlets gained popularity in the 1970s and is usually credited to Bradbury and Khadem [106] with review summaries in Knowles and Saddlington [107] and more recently in Zaman, Bridges, and Huff [108]; however, there was insufficient literature about tabs in diffusers. Literature did exist for implementing ‘vortex generators’ in diffusers—that was investigated in the following section—but these devices are better described as flow-directing vanes since they do not provide a bluff body against the flow.

Tabs for nozzle applications have been dubbed as “super mixers” since they have the potential to considerably increase the jet spreading rate but typically incur a minimal thrust penalty. Numerous studies [109–115] investigated the effect of tab geometry and orientation on performance or jet-plume development. A general consensus was summarized using Zaman, Reeder, and Samimy [109] who studied square (simple) and triangular (delta) tabs for jet noise suppression (with additional contributions [116–118]). Results showed that:

i. Compressibility had little significance on the interaction of streamwise vortices.

ii. Tab shape had no noticeable difference in the amount of distortion produced where round, square, and triangular were tested but delta tabs performed slightly better. For example, delta tabs that each occupied 1.5–2% of the nozzle outlet with a $90^\circ$ apex were observed to produce streamwise vorticity because of a substantial pressure differential between the upstream and downstream sides of the tab and the tab sides shed vortex filaments with fewer losses.
iii. Tab width was more influential on distortion than tab height for simple tabs provided that the tab height was comparable to or exceeded the boundary layer thickness. Tabs that did not protrude into the core flow did not produce any observable indentations.

iv. Tab orientation was more influential with the most pronounced effect occurring for tab orientation angle of $\phi = 135^\circ$ except Park et al. [114] who found that $\phi = 90^\circ$ tabs (tabs oriented normal to the flow) were the most effective in reducing reattachment length downstream of a backward facing step.

v. Four delta tabs were preferable for providing the greatest amount of mass flux with downstream distance of the exit since they created four indentations that stretched the mixing layer area in comparison to none, two and six (only three indentations) tabs. Ahuja and Brown [110] and Behrouzi and McGuirk [113], however, concluded that two tabs performed better in their respective screech noise and potential core reduction studies.

vi. Tabs placed slightly downstream from the nozzle end became less effective.

**Diffuser Applications**

Wood [119] conducted an investigation on the effectiveness of vortex generators on a straight outer wall $R/R_o = 0.69$ short annular diffuser with conical centre-body. Addition of the rectangular non-cambered NACA 0012 airfoil VGs oriented normal to the walls reduced fluctuations and improved static pressure recovery. Vortex generator pairs were set at opposite angles of attack to produce counter-rotation. Results were found to be sensitive to the tested range of Mach numbers up to 0.55. The best arrangement located 24 VGs on the inner wall at the centre-body base with a span (height) of $0.15\Delta R_t$ and 44 smaller VGs just downstream of the diffuser inlet with reduced influences associated to a $15^\circ$ angle of attack (incidence) and chord length. It was observed that VGs on the inner wall always improved pressure recovery when compared to the bare configuration whereas the addition of VGs on the outer wall minimally increased the effectiveness an extra 2% at low speeds. It
was suggested that the benefits of including a preferential VG configuration on the outer wall would be small.

Valentine and Carroll [120,121] tested VGs in short wide-angle conical diffusers. Preference was given to the counter-rotating configuration that placed adjacent NACA 0012 airfoils with opposite angles of attack, one-third to one-half the inlet diameter upstream of the diffuser inlet; pressure recovery significantly increased over the non-augmented configuration. The recommended span height for the airfoils was six times the displacement thickness (on the order of $\delta$) but did not appear to be a critical quantity when compared to spans of 4- and 10-times $\delta^*$. The investigation also concluded that an angle of attack be between $14^\circ$ and $20^\circ$ and adjacent airfoils be separated by a space of two spans. Further improvements occurred for decreasing inlet boundary layer thickness. Placing the VGs further upstream was favourable but seen as impractical since it extended the diffuser length.

Stevens and Williams [34] investigation of a straight-core annular diffuser found that turbulence was promoted from both coarse screens and turbulence generators (spoilers) and resulted in increased static pressure recovery with approximately no negative impact on total pressure. Two diffusers with $R_i/R_o = 0.83$ were designed that used Sovran and Klomp’s [1] performance maps to obtain maximum pressure recovery at the prescribed non-dimensional length and area ratio respectively. Maximum pressure recovery and a stable outlet flow occurred when the spoilers were located $7D_h$ upstream of the diffuser inlet on the outer wall. Improvement was attributed to the radial transfer of momentum that improved the turbulent shear stress distribution and reduced the exit blockage. It was observed that the velocity gradient at the wall was $\partial \tau/\partial y \neq \partial p/\partial x$, implying that advection terms were significant.

Sullerey, Mishra, and Pradeep [122] were interested in reducing exit flow distortion and improving total pressure recovery in two-dimensional S-duct diffusers of different radius ratios that had the same streamwise length. Fences placed on wall centrelines and two types of vortex generators were tested: a wishbone type whose purpose was to energize the boundary layer in regions of high
adverse pressure gradients and a tapered-fin type that controlled secondary flows. Measurements that estimated $\delta$ and severity of adverse pressure gradients were taken on a bare diffuser to help select the fence configurations and VG locations. The diffuser with the less severe S-bend radius developed greater pressure recovery and the fences with an optimum height of $1.2\delta$ resulted in a 9% improvement over the best VG configuration. In comparison, tapered-fin VGs out-performed fences by 7% in the sharper-bend diffuser and the wishbone type VGs only gave marginal improvement.

Similar studies were completed by Reichert and Wendt [123].

Kaldschmidt, Syltebo, and Ting [124] experimented with several types of flow control devices—airfoils, fences, and turning vanes—in an S-duct. Tests occurred with VGs positioned on both upper and lower surfaces in the vicinity of the convex curvature. Airfoil VGs that generated co-rotating vortices were preferred since the induced vortices remained closer to the wall with lower loss to the inlet pressure. Improvements were not observed with the fences or turning vanes.

Jirásek [111, 76] performed a numerical analysis with experimental validation on airfoil-type flat rectangular-shaped VGs, similar to those shown in Fig. 2.8, that determined preferential configurations in an S-duct with diffuser. Five variables were defined: height $h$, length $l$, spacing $s$, upstream distance from the separation point $x_s$, and inclination angle $\phi$. Dominant variables were $h = 0.3\delta$ and $x_s$ located upstream of the separation point. Relatively weak dependencies were observed for $l$ and $s$, and pressure distortion was independent of $\phi$. The procedure developed solutions with substantially reduced flow distortion and improved pressure recovery. Wind tunnel tests measured lower pressure recovery but of similar magnitude to CFD predictions with the VGs, likely because additional vortices were observed.

Vortex Generator Models

Modelling geometrical effects with equations characterizing the disturbance that causes the formation of vortices was an attractive opportunity that received increasing attention in recent years. In particular, potential flow solutions were adapted to simulate the effects of vortex generating airfoils.
Two models were developed: (i) the vortex-source model and (ii) the lifting-force model.

Vortex-source models construct a source term based on a free vortex of circulation $\Gamma$ that is suddenly generated where the VG is located. The velocity profile caused by the VG is expressed by something similar to (74 expressions given that modelled NACA0012 wing sections):

$$u = \frac{A}{x} \exp \left( -\frac{U_\infty r^2}{4vx} \right)$$  \hspace{1cm} (2.39)

$$v_\theta = \frac{\Gamma}{2\pi r} \left[ 1 - \exp \left( -\frac{U_\infty r^2}{4vx} \right) \right]$$  \hspace{1cm} (2.40)

$$v_r = -\frac{Ar}{2\pi^2} \exp \left( -\frac{U_\infty r^2}{4vx} \right)$$  \hspace{1cm} (2.41)
The integration constant \( A = \frac{D_0}{4\pi \rho v} \) for VG profile drag \( D_o \) is found by equating the change of momentum in the VG wake to the VG drag. Circulation is a function of the VG geometry,

\[
\Gamma = C_\alpha \frac{C_L}{2} c_l u
\]

where \( C_L \) is the lift coefficient, \( c_l \) is the VG chord length, and \( u \) is the flow velocity at the VG tip. \( C_\alpha \) is a constant that considers viscosity and turbulence effects; from inviscid wing theory and experiment, \( C_\alpha \) cannot be greater than 0.45.

Similarly, Wendt’s [12] vortex-source model simulates vortices shed from wall-mounted vane-type generators. The strength of each vortex is a function of the VG chord length, height, angle of incidence, local freestream velocity, and boundary layer thickness. Circulation of the shed tip vortex is based on Prandtl’s inviscid airfoil theory. Density, momentum, and energy values are defined as functions of circulation. The model was implemented by Dudek [91] whose simulations of VGs in a pipe and S-duct predicted similar trends with varying degrees of agreement to experiment.

von Stillfried and colleagues [125–127] developed a statistics-based vortex-source model. An additional equation modelled the second-order statistics (vortex stresses) as Reynolds stresses. Circulation was integrated into the momentum equation as a drag force that implemented a windowing function to yield a smooth profile on the cells encompassing the VG. A modified RSM model modelled the pressure-strain rate. In [126], flat-vane VGs were tested in a counter-rotational setup over various angles of incidence. Flow separation was successfully predicted in both 2D and 3D simulations; however, the forcing as opposed to the developing nature of the vortex stresses resulted in slower and weaker shear stress components than those measured in experiment.

Jirásek [128] proposed a lifting-force model that estimated a lift force from the Prandtl lifting theory and noted that it was a more promising model since it did not require initial user input parameter estimates. The model added a source term to the momentum equations, so it was necessary to apply the VG model on cells with volume \( \Delta V_i \). Jirásek’s jBAY model was a modification to the Bender-Anderson-Yagle (BAY) model that defined a source term \( L_i \) dependent on the airfoil
2.4. DIFFUSER AUGMENTATION

planparallel area $S_{VG}$ and angle of attack $\alpha$:

$$L_i = C_{VG} S_{VG} \sum V_i \Delta V_i \rho \alpha u^2 l$$  \hspace{1cm} (2.43)

The modification assumed a zero thickness VG—a simplification for defining the model control points. The constant $C_{VG}$ is a relaxation parameter that controls the strength of the side force, and $\alpha = u \cdot n/|u|$ and $l = u/|u| \times b$ for normal $n$ and span $b$ unit vectors. The density and velocity are interpolated from the cell nearest to the defined VG and the resulting side force is redistributed back to the nodes. This method is not completely grid independent since sufficient grid resolution is required in the region just downstream of the forcing region where the generated vortex structures need to be resolved [126].

Incorporating a VG model in CFD requires representation of the VG with a control volume. Dimensions are typically selected to enclose the VG with a rectangular volume. Implementing a VG model in a numerical optimization study greatly reduces the amount of time to generate grids and reduces the number of grid points surrounding a VG because the actual geometry is not resolved. Since VGs are typically placed within the boundary layer, the grid density is already sufficiently dense to be capable of resolving the vortex filaments shed by the VGs [91, 126, 128].

Fernández et al. [129] compared different methods of accounting for a single airfoil-type VG at 20° angle of attack on a flat plate: grid-resolving the VG, a lifting-force model, a vortex-source model analytical solution (the simplicity of the problem did not require CFD), and experimental data. The implemented VG model included an algorithm that only applied forces to cells within the outline of the VG geometry. The computational effort of the VG model was about 0.1% of that required to grid-resolve the VG, mainly because of the meshing reduction since the VG boundary layer was not resolved. Good agreement of axial and azimuthal velocities occurred between experimental and analytical; however, the CFD results were not as well-predicted with the mesh-resolved solution being about 10% closer to experiment.

It was clear that VG models showed tremendous promise for design purposes in CFD; however,
further improvement was necessary to increase the accuracy (without having prior knowledge of the solution). Since physically modelling a vortex generator appeared to provide better comparison to experiment, a VG model to simulate the effect of tabs was not developed for this project.

### 2.4.2 Other Methods

Other augmentation methods have been evaluated to improve diffuser performance; however, the literature either reported performance penalties due to their implementation or the method was not as practical in comparison to vortex generators to retrofit to a turbomachinery diffuser.

**Suction**

Suction maintains a constant low velocity level along the diffuser wall where results reported that bleeding approximately 6% improved effectiveness to desirable levels \[130\]. Problems with this method include a weight penalty since a pump is necessary to extract the fluid from an exhaust diffuser and the heat signature associated with the suction exhaust.

**Blowing**

Wall injection is capable of energizing the boundary layer but may not totally remove flow separation. Blowing does not necessarily have to be forced and was always found to be beneficial \[5\]. Henry and Wilbur \[131\] determined that > 4% injection yielded a 40% increase in pressure recovery and produced more improvement in the exit velocity profile than suction control. For annular diffusers, the challenge is in entraining fluid into the inner cylinder to energize the flow near the centre-body.
2.4. DIFFUSER AUGMENTATION

Wall Roughness

The friction created by adding wall roughness (similar for ribs or fins) can offset the wall shear decrease caused by an adverse pressure gradient. Persh [132] found that no flow separation was observed at the diffuser exit for any amount of roughness (cork particles) applied to the surfaces whereas the same 23° conical diffuser with smooth walls had substantial separated regions. In some cases, total pressure loss was found to decrease (improve); however, static pressure recovery decreased as the amount of surface roughness increased.

Screens

Screens placed across the flow passage are capable of preventing separation or causing separated flow to reattach, thus improving the flow field uniformity and stability. Mechanisms include increasing the normal velocity gradient near a surface and decreasing the pressure gradient. Screens incur a static pressure penalty so it was suggested that wind tunnels are the best application since reduced turbulence and high flow uniformity are desired [5].

Vanes or Splitters

The principle of insert vanes is to subdivide the diffuser flow along naturally occurring streamlines to achieve more optimum loading. Feil [133] suggested that the number of vanes used should form individual passages with expansion angles of 7–10°. An optimum vane length can be selected from charts based on the passage area ratio. For diffusers with divergence angles between 40° and 80°, adding vanes increased pressure recovery and prevented stall. Working with this principle, Welsh [134] tested an insert star that used eight cylindrical spokes and found for a 11° half-angle conical diffuser with an inlet blockage of 2% that the best performance occurred for stars located 1.9 inlet radii downstream of the inlet plane, overall diameter equal to the inlet diameter, and sized to give a flow blockage of 10–14%.
A distributed exhaust nozzle that consisted of expanding air through many small mini-nozzles was a noise reduction concept investigated by Kinzie et al. [135]. In comparison to experiment, CFD studies that used the SST model slightly over-predicted the potential core length but obtained good jet axial velocity decay. Computational results showed no evidence of mini-jet identity at the outlet plane—a feature required to achieve noise reduction—since the flow through the mini-nozzles quickly coalesced into a single plume. Experimental results confirmed that noise reduction, based on data obtained from a linear microphone array, did not occur. Recommendations proposed that coalescing can be prevented if the mini-jets were spread further apart.

Alternatively, splitters can significantly change the natural flow path. Yang [136] oriented a ring insert to modify the loading along the wall that improved pressure recovery. It was observed that inlet velocity and turbulence intensity were secondary effects to the induced velocity caused by the splitter. Sajben et al. [137] performed a parametric study over a range of conical diffuser lengths that inserted a flow control rail (FCR) and established three design rules to reduce shape factor and enhance pressure recovery: (1) the FCR thickness should equal the boundary layer thickness; (2) it should be placed parallel to the wall and \( h = 4\sqrt[3]{\delta} \) from the wall; and (3) be \( 5h-7h \) upstream of the separation point in a bare diffuser.

### 2.4.3 Augmentation Review Summary

Table 2.1 lists the diffuser lengths and area ratios evaluated from several augmentation device reviews. For annular diffusers, the divergence angle was calculated from its conical equivalent where radius was first determined from \( r = \sqrt{A/\pi}. \) The diffusers were noted as being designs that achieved maximum pressure recovery at the prescribed non-dimensional length and/or had aggressively wide angles that resulted in appreciable stall absent of any augmentation devices. Pressure recovery was improved by as little as 13% (bare diffuser effectiveness of 80% [137]) to 40% (bare diffuser effectiveness of 55% [121]). (Aside: the cited literature used different equations for defining static pressure coefficient \( C_p \); therefore, the reported \( \eta \) values are subject to interpretation.)
2.4. DIFFUSER AUGMENTATION

### Table 2.1: Tested augmentation device review geometries

<table>
<thead>
<tr>
<th>Reference</th>
<th>Diffuser type</th>
<th>$L/\bar{r}$, $^a$</th>
<th>$AR$, $^a$</th>
<th>$2\theta_d$, $^a$</th>
<th>$AR$ at max $C_p$, $^b$</th>
<th>$AR$ at 1st stall$^b$</th>
<th>Improvement in $\eta$ with VG, %</th>
</tr>
</thead>
<tbody>
<tr>
<td>Wood [119]</td>
<td>annular</td>
<td>6.9</td>
<td>1.9</td>
<td>15</td>
<td>2.2</td>
<td>2.3</td>
<td>21</td>
</tr>
<tr>
<td>Valentine and Carroll [120, 121]</td>
<td>conical</td>
<td>2.0</td>
<td>2.0</td>
<td>23</td>
<td>1.5</td>
<td>1.8</td>
<td>40</td>
</tr>
<tr>
<td>Stevens and Williams [34]</td>
<td>annular</td>
<td>5</td>
<td>2.0</td>
<td>31</td>
<td>2.0</td>
<td>2.1</td>
<td>30</td>
</tr>
<tr>
<td>Sullerey et al. [122]</td>
<td>$AS = 0.8$ S-bend$^d$</td>
<td>1.7</td>
<td>1.35</td>
<td>5.6</td>
<td>1.6</td>
<td>1.6</td>
<td>19</td>
</tr>
<tr>
<td>Sajben et al. [137]</td>
<td>conical</td>
<td>1.55</td>
<td>2.83</td>
<td>22</td>
<td>1.4</td>
<td>1.7</td>
<td>13</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td>1.55</td>
<td>4.76</td>
<td>13</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Welsh [134]</td>
<td>conical</td>
<td>5.1</td>
<td>2.0</td>
<td>22</td>
<td>1.9</td>
<td>2.5</td>
<td>26</td>
</tr>
<tr>
<td>App. [11, 12]</td>
<td>annular</td>
<td>12</td>
<td>1.91</td>
<td>10</td>
<td>3.3</td>
<td>2.8</td>
<td>-30</td>
</tr>
</tbody>
</table>

---

$a$ conical: $\bar{R} = r_t$, annular: $\bar{R} = \Delta R_t$, channel: $\bar{R} = W_t$

$b$ conical values from McDonald and Fox [138], annular maximum $C_p$ from Sovran and Klomp [1] and first stall from Howard [4], and channel values from Reneau et al. [139]

$c$ at prescribed area ratio

$d$ assumed straight square channel for analysis

Numerous investigations have been completed that studied the effects of different types of passive flow control devices on diffuser performance. Parametric studies allowed for the identification of preferable geometries and configurations with good success. An apparent gap in literature existed that considered (i) implementing tabs in diffusers and (ii) studying the effects caused by VGs in the presence of a swirling inlet flow. Tabs were much easier to manufacture and retrofit to existing systems, than airfoils for example, since there were looser tolerances associated with the alignment and precision with respect to the adjacent VGs. Typical aircraft operating conditions have swirling exhaust flows so knowing the VG performance in unswirled flow was not always useful.
Chapter 3

Experimental Methods

This chapter describes the resources used to obtain and analyze the experimental data:

3.1 Experimental apparatus description

3.2 Instrumentation and data reduction

3.3 Data assessment

3.1 Experimental Apparatus

Tests were completed on a cold flow wind tunnel located in the Queen’s University Gas Turbine Laboratory. Figure 3.1 shows a 28 kW (37.5 BHP) centrifugal blower constructed by New York Blower Co. that delivered air at up to 13 kPa of static pressure at 3550 rpm. The blower was driven by a 30 kW (40 HP), 575 V, 36 amp Baldor Industrial electric motor that rotated at 3530 rpm and controlled by an SMVector variable speed drive manufactured by Lenze - AC Tech. A butterfly valve manufactured by Valvtech Inc was located just downstream of the blower outlet and had experimentally observed loss coefficients of 2, 3, 26, 83, and 410 when the valve was respectively fully open, $\frac{3}{4}$, $\frac{1}{2}$, $\frac{3}{8}$, and $\frac{1}{4}$ open.
3.1. EXPERIMENTAL APPARATUS

Figure 3.1: Wind tunnel schematic (units in millimetres).

Downstream of the valve, the flow path transitioned from a round to a square cross section and was diffused prior to entering the settling chamber. The flow passed through vanes, screens, and filters intended to straighten the flow and reduce flow turbulence. The internal corners of the settling chamber were chamfered to create an octagonal shape designed to reduce corner effects. Lastly, the flow was accelerated through a bell mouth intake and provided a maximum uniform flow of 2.9 kg/s to a $D_o = 152.9$ mm (6 in) diameter cross section.

Figure 3.2 is a photograph showing the downstream face of the settling chamber with annulus and test section attached. An existing annulus, with an inner length of 1.4 m (0.4 m projected into the bell mouth) and diameter of 81.0 mm (0.313 $D_o$), was attached to the bell mouth. The outer cylinder was commercially-available acrylic round pipe. The concentricity of the annulus height at the outlet was $\Delta R_s = 36.0 \pm 0.6$ mm where the centre of the inner cylinder aluminum pipe was offset towards the bottom-right corner (SE). The inner cylinder had an upstream ellipsoidal bullet and was supported using 2x4 NACA 0020 35 mm-long airfoils that were equally spaced clockwise circumferentially at 70°, 160°, 250°, and 340° from the z-axis and axially located 660 mm apart with the second set located 220 mm upstream of the annulus outlet.

Based on four total pressure probes with three located between airfoils and one directly upstream
of an airfoil strut that traversed a cross section near the annulus inlet, all four probes showed that at least 75% of the flow velocity was constant with an inner boundary layer $\delta_i < 0.09\Delta R_s$ and outer boundary layer $\delta_o < 0.15\Delta R_s$. Flow entering the annulus was uniform with an area blockage of $A_{B,up} = 0.022–0.052$.

Figure 3.3 is a contour plot showing an annulus outlet traverse (with downstream ducting removed). The contours show that reasonable symmetry of $\pm8\%$ in the axial flow was obtained that was influenced by the airfoil struts; however, the vectors, particularly in the top two quadrants, identified that the flow was not axisymmetric as there was an apparent cross-flow, from left to right, on the order of $3\%$ of the axial flow.

Flow area blockage at the annulus outlet (section 5 at $x = -1.05D_o$) was on average $A_{B,s} = 11\%$. Given that the annulus had a length of 20 hydraulic diameters, this was just short of fully-developed flow based on Jonsson and Sparrow’s [28] experiments. Assuming $\Delta R_s = 2\delta$ and a turbulent flow $1/7$th power-law velocity profile gave $A_{B,s} = 12.5\%$; this larger value corroborated that the experimental annulus outlet profile was almost, but not quite, fully-developed.

Figure 3.4 is a schematic of the test section affixed to the bell mouth intake. A transition duct was attached to the annulus that increased the inner diameter from $0.53D_o$ to the annular diffuser inlet inner diameter of $114.3$ mm ($D_i = 0.75D_o$). The inner cone of the transition duct was 102
3.1. EXPERIMENTAL APPARATUS

Figure 3.3: Annulus outlet total pressure contours, non-dimensionalized by the mass-flow averaged total pressure $p_{0,s}$.

mm long with a radial concentricity relative to the annulus inner cylinder axis of ±0.3 mm. A throat length of 50.8 mm (0.33D_o) machined from a solid aluminum bar was provided prior to flow entering the annular diffuser.

Swirlers or a spacer were attached directly onto the end of the annulus at the swirl station noted in Fig. 3.4. Three straight-vaned swirlers consisting of 16 equally spaced flat blades, 1.9 cm long were used with nominal angles of $\vartheta = 10^\circ$, 20°, and 40°. A curved vane (CV) swirler was also tested—the flat vanes were twisted along the span.

Swirl angle profiles evaluated using Eq. (2.18) are shown in Figs. 3.5(a)–(b). Axial and tangential velocity profiles for the swirlers are in App. A.1. Actual flow angles and swirl numbers are given in Tab. 3.1. Although the average profiles were approximated as linear, the left and right-side radial traverses identify that the individual blades had local bias.
CHAPTER 3. EXPERIMENTAL METHODS

Figure 3.4: Test section schematic depicting instrumentation locations. Section 5 = 3-hole probe traverse at annulus outlet ($x = -1.05D_o$) and section 1 = 3-hole probe traverse at annular diffuser inlet ($x = -0.23D_o$). Red squares denote wall pressure taps. At least two instruments were placed at each axial location.

Figure 3.5: Traversed swirl profiles (downstream ducting removed). Linear extractions of left (W) and right (E) sides included.

3.2 Data Acquisition

Instrumentation used to collect pressure data included 1.6 mm outer diameter Scanivalve wall pressure taps, and three- and seven-hole pressure probes that were manufactured in-house. The 3-hole
Table 3.1: Swirler analysis from traverses at the annulus outlet \((x = -1.05D_o)\) and annular diffuser inlet \((x = -0.23D_o)\) with downstream ducting removed (positive CCW).

<table>
<thead>
<tr>
<th>Swirler Nominal Angle</th>
<th>section (5) Actual Angle, (\vartheta_s, ^\circ)</th>
<th>Swirl Number, (S_s)</th>
<th>section (1) Actual Angle, (\vartheta_t, ^\circ)</th>
<th>Swirl Number, (S_t)</th>
</tr>
</thead>
<tbody>
<tr>
<td>S0</td>
<td>-0.6</td>
<td>-0.02</td>
<td>-0.3</td>
<td>-0.03</td>
</tr>
<tr>
<td>S10</td>
<td>9.6</td>
<td>0.28</td>
<td>4.5</td>
<td>0.28</td>
</tr>
<tr>
<td>S20</td>
<td>20.6</td>
<td>0.62</td>
<td>11.5</td>
<td>0.71</td>
</tr>
<tr>
<td>S40</td>
<td>42.2</td>
<td>1.55</td>
<td>28.6</td>
<td>1.92</td>
</tr>
<tr>
<td>SCV</td>
<td>-8.9</td>
<td>-0.27</td>
<td>-5.1</td>
<td>-0.32</td>
</tr>
</tbody>
</table>

probe had a 60\(^\circ\) chamfered head 1.3 mm high and 3.8 mm wide and the tip was 14 mm upstream of the shaft. Three-hole probe traverses occurred at section (5) (annulus outlet) located at 161 mm- \((x = -1.05D_o)\) and section (1) (annular diffuser inlet) located at 35 mm- \((x = -0.23D_o)\) upstream of the actual annular diffuser inlet at plane DS. The 7-hole probe consisted of a 3.9 mm diameter rounded tip with each pressure port having a 0.406 mm diameter. The probe stem had a 6.4 mm diameter and 100 mm length. The probe tip was located 9 mm downstream and centred with respect to the diffuser outlet.

Measurement locations of the various instruments are shown in Fig. 3.4 where at least two instruments were placed at each axial location in the diffuser. Figure 3.6 is a picture taken during an outlet traverse that looks upstream into the test section at the diffuser outlet. The tips of the 3-hole probes remained in the flow against the outer wall. All experiments were completed with one row of tufts on the north face of the CB whereas extra tufts were added for some configurations. The typical orientation of the wall pressure taps is shown in Fig. 3.7. Additional OW taps (dashed lines) were located in the SE and NW quadrants roughly at \(x = 0.3, 0.7, \text{ and } 1.2D_o\). The 3-hole probes were inserted from W and E.

The three-hole probes were designed and calibrated to provide axial and tangential velocity components in addition to static and total pressures. Three-hole probe calibration and reduction methodology is outlined in App. A.2.
Figure 3.6: Upstream view of instrument locations (Config. bd shown)

Figure 3.7: Wall pressure tap orientation (looking upstream).
3.2. DATA ACQUISITION

The seven-hole probe measured all three velocity components. The implementation of the polynomial curve-fit calibration procedure developed by Gallington [140], that was similar to procedures used to reduce 3-hole data, is described in Crawford and Birk [141–143]. Additionally, data were collected from two Omega Type K thermocouples: one located on the seven-hole probe just downstream of the tip and the other inserted near the annulus inlet in the centre of the annulus gap.

Other instrumentation, such as hot-wire anemometry or particle image velocimetry, may give a better flow description; however, pressure probes were much more robust in their ability to conveniently obtain reliable data under most flow conditions. Although performance was weakly dependent on turbulent statistics, larger contributions were associated with geometry and global fluid properties: sufficient design guidance and CFD validation was provided using the pressure measurement instruments that calculated the first-order statistics of interest.

Pressure was detected through Tygon tubing connected to the Omega 1 psi (6.9 kPa) differential pressure transducers that required a regulated 5 V excitation voltage and delivered an analog signal in the output span of 0.25 to 4.25 volts [144]. The transducers were calibrated assuming a linear curve-fit to numerous manometer readings over the full scale. To eliminate bias due to a possible difference between the pressure transducer calibration atmospheric pressure and experimental conditions, offset pressures were added to the measurements. The pressure transducer sampling analysis is in App. A.3.

Data were collected on a Dell Optiplex GX620 computer with a Pentium 4 processor running at 3.2 GHz with 3 GB of RAM from a Data Translation Inc DT3003-PGL data acquisition (DAQ) board with a DT730-T terminal block. The DAQ card was capable of accepting 32 differential analog inputs, each with 12-bit resolution [145]. Unused channels were grounded to reduce noise and cross-talk.

Outlet measurements were obtained by mounting the seven-hole probe on an X-Y traverse table. Two six-wire stepper motors, manufactured by Applied Motion Products Inc, required a DC power supply of 4.7 volts at 1.8 amps to provide 200 steps per revolution. Motor motion was achieved
by receiving signals from the L702N motor control board that was connected to the computer’s parallel port. The horizontal motion was belt-driven and moved 0.255 mm/step whereas a worm drive controlled the vertical motion at 0.0254 mm/step.

Traverses were completed using Cartesian coordinates with a user-supplied uniform increment for both directions. Traverses were always assigned to end at the origin to confirm proper incrementation; however, a discrepancy of ±1 mm was occasionally observed along the horizontal axis. The probe always returned to the proper vertical location.

### 3.2.1 Fluid Properties

Experiments were conducted using atmospheric cold-air. The bulk of tests were completed over a hydraulic diameter Reynolds number range at the annular diffuser inlet of $0.8 \times 10^5 < \text{Re}_t < 2.3 \times 10^5$ with corresponding Mach numbers of $0.10 < M_t < 0.28$. Additionally, several tests were completed upwards of $\text{Re}_t \approx 3.5 \times 10^5$ and $M_t = 0.44$. Incompressible flow was assumed for all experimental tests. Air flow temperature within the duct was usually 1°C above ambient.

### 3.3 Data Confidence

Defining acceptable tolerances associated with assembling experimental apparatuses is a challenge. The absence of perfection in the real-world requires engineering judgment to identify a tolerable amount of imperfection in assessing the reliability of experimental data. Kline and McClintock’s [146] discussion of experimental errors led to the proposal to have a quantifiable description of the uncertainty. Since the inception of reporting results using confidence intervals, authors including Moffat [147, 148], Benedict [149], and Beckwith, Marangoni and Lienhard [150] have elaborated the contributions and procedure for conducting an experimental uncertainty analysis. The root-sum-square (RSS) model became widely accepted in quantifying total uncertainty.

Although geometry errors can be included as a bias in the uncertainty equation, experimentalists
may lack the necessary tools and resources to obtain the required accuracy. For example, evaluating the concentricity of an annulus or confirming axis alignment to fractions of a millimetre when connecting multiple ducts. In studies where resolution is critical such as determining the reattachment point downstream of a backwards facing step [151] or quantifying higher-order statistics in turbulent channel flow [152], tight tolerances are essential. However, for analyses with first-order statistics objectives, larger geometrical tolerances may be acceptable.

Literature relating to the experimentation of diffusers rarely explicitly states discrepancies relating to the geometry. Tolerances can be inferred from lengths, that were often reported with an accuracy of three significant digits, and axial velocity profiles, that for example were symmetric in conical diffusers to within ±1% [153] and ±5% [154]. This information is of little practical use in the aerospace industry where aftermarket exhaust systems, often manufactured from sheet metal, may have larger geometrical tolerances. Moreover, even if the exhaust system is perfectly aligned with the aircraft’s turbomachine, it is unlikely to maintain alignment after repeated use.

Measurement uncertainty procedures are included in App. A.4. Table 3.2 lists chosen configuration (CB|OW, see Fig. 5.1 for configuration geometry) uncertainties of the performance quantities and selected variables. Diffuser inlet uncertainties (section 1) measured by the 3-hole probes were, on average, within 1.6% for all configurations. Outlet axial velocity and total pressure uncertainties measured by the 7-hole probe were characteristic of the chosen outer wall component. Greater uncertainties were associated with $C_p_0$ due to non-dimensionalizing the absolute uncertainty by a small value and with $\gamma$ because of chaotic and/or reversed flow at the outlet plane.

Biases were present during experimentation and scrutinized in App. A.5. Provided that any misalignment was unintentional, the biases were not significant enough to discredit the results. For example, the 7-hole probe was usually out of alignment by 0.6° in pitch and 1.3° in yaw with respect to the rig axis; this discrepancy was not significant enough to lose confidence in the calculated axial velocities.
Table 3.2: Performance quantity uncertainty of selected configurations (CB|OW, see Fig. 5.1 for geometry) that included outlet auxiliary experiments with the given amount of swirl.

<table>
<thead>
<tr>
<th>Config.</th>
<th>Swirl</th>
<th>Uncertainty, % (95% CI)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td>$C_b$</td>
</tr>
<tr>
<td>aa</td>
<td>S0</td>
<td>1.8</td>
</tr>
<tr>
<td>bb</td>
<td>S0</td>
<td>1.7</td>
</tr>
<tr>
<td>cc</td>
<td>S0</td>
<td>1.7</td>
</tr>
<tr>
<td>bb</td>
<td>S10</td>
<td>1.7</td>
</tr>
<tr>
<td>bb</td>
<td>S20</td>
<td>1.7</td>
</tr>
<tr>
<td>bb</td>
<td>S40</td>
<td>3.8</td>
</tr>
<tr>
<td>bb</td>
<td>SCV</td>
<td>1.7</td>
</tr>
<tr>
<td>bd</td>
<td>S20</td>
<td>2.1</td>
</tr>
<tr>
<td>bb 4 tabs</td>
<td>S20</td>
<td>1.8</td>
</tr>
<tr>
<td>bc 4 tabs</td>
<td>S20</td>
<td>2.2</td>
</tr>
<tr>
<td>bd 4 tabs</td>
<td>S20</td>
<td>2.0</td>
</tr>
</tbody>
</table>
Chapter 4

Numerical Setup

The commercially available computational fluid dynamics code ANSYS Fluent ® (releases 13.0, 14.0, and 14.5) [84] was used to obtain steady-state solutions on modest finite volume structured grids with the Reynolds-averaged Navier-Stokes equations and two-equation turbulence models. Preference was given to the realizable $k$-$\varepsilon$ turbulence model in the optimization study since it was regarded by some [88] as being the preferred turbulence model for diffusers. Analyses were also completed comparing results obtained with the standard- and RNG-$k$-$\varepsilon$ and the SST turbulence models. Two grids were developed in recognition of the different wall treatment methodologies.

4.1 Computational Domain

Figure 4.1 shows the computational domain for the diffuser systems. Three blocks were modelled: transition duct, annular diffuser, and outlet plenum. The transition duct was included to accommodate comparative analysis at section (1) ($x = -0.23D_o$) and to allow some flow development of the unknown turbulent quantities. A plenum control volume was included downstream of the diffuser outlet for numerical computations to resolve any gradients, such as flow reversal, that may extend beyond the outlet plane. For three-dimensional grids, symmetry planes were assigned with either symmetry (no swirl) or periodic boundary conditions.
4.1.1 Boundary Conditions

The fluid was modelled as compressible air using the ideal gas law where $MW = 28.966$ kg/kmol. Sutherland’s law modelled viscosity:

$$\mu = \mu_0 \left( \frac{T}{T_{ref}} \right)^{3/2} \frac{T_{ref} + Su}{T + Su}$$  \hspace{1cm} (4.1)$$

where $\mu_0 = 1.716 \times 10^{-5}$ kg/m-s, $T_{ref} = 273.11$ K, and $Su = 110.56$ K. Specific heat capacity was defined by a piecewise-polynomial function of temperature with the coefficients for air provided from the Fluent database. For all CFD, walls were defined as adiabatic with no slip. Coarse grids simulated roughness heights of 2 $\mu$m–10 $\mu$m assuming the walls were roughened with tightly packed, uniform sand-grains. No change in performance was observed with respect to no slip walls (if unstated, performance refers to the coefficients $C_b$, $\gamma$, and $C_{\rho_0}$).

Plenum boundaries were specified with quiescent atmospheric air at $T_0 = 300$ K, 1% turbulence.
4.1. COMPUTATIONAL DOMAIN

intensity, and turbulence length scale \( l = 0.4\delta \) for the wall-bounded flow where \( \delta \approx 0.001 \) m was measured on the plenum wall in a preliminary simulation. Pressure inlets were specified on the front and top surfaces whereas a pressure outlet was assigned to the downstream surface.

It was not as straightforward to define the transition duct inlet boundary condition. In general, care must be taken since incorrect boundary conditions can lead to the wrong conclusions when evaluating turbulence model performance against experimental data. Having said that, there were challenges in obtaining turbulence data from experiment: \( k \) can be measured but no satisfactory methods were available to quantify inlet \( \varepsilon \) [83]. Results must be scrutinized where, for example, larger inlet length scales should lead to shorter reattachment lengths over a backward facing step. Swirling flows are quite sensitive to inlet boundary conditions and the overall accuracy is dependent on the grid quality [155].

For the optimization study undertaken in App. B where CFD served as a design tool to study geometric alterations, a realistic representation was not required. A transition duct was not modelled. A uniform unswirled mass-flow inlet was specified with constant total temperature of 800 K, \( Re_{inlet} = 8.5 \times 10^4 \), and \( M_{inlet} = 0.23 \) to mimic typical gas turbine exhaust conditions. Turbulence properties were assumed as 5% turbulence intensity with turbulence length scale scaled by the hydraulic diameter \( l = 0.07D_h \) to represent fully-developed internal flow.

Since a purpose of the CFD, presented in Ch. 6 and supplemental results appendices, was for comparison purposes to experiment, measured properties at section 5 were supplied to the CFD transition duct inlet. Two types of inlet boundary conditions were available and applicable for the implemented compressible flow equations:

- a pressure inlet boundary condition that allowed mass flux to vary
- a mass flow inlet boundary condition that permitted total pressure to vary in response to the interior solution

The pressure inlet condition was selected since it was more conservative (slower convergence) and
recommended in the user manual \cite{84}. The inlet boundary defined at section 5 was supplied with an experimentally obtained total pressure profile and for swirl simulations, the tangential velocity component profile.

To determine if the weather conditions during the experiments had any influence, sensitivity CFD was completed on the defined operating conditions. Based on a coarse 2D grid analysis without swirl, simulations were completed over a total temperature range of 290–310 K. In comparison to a solution with $T_0 = 302$ K, change in performance was below 0.7%; therefore, a constant $T_0 = 302$ K inlet total temperature was specified throughout. A similar study was completed on the specified atmospheric pressure over the range of 98 kPa to 105 kPa. In comparison to a solution with $p_{atm} = 1$ atm, performance stayed within 0.3%, so the reference static pressure operating condition remained unchanged at 1 atm.

Turbulence quantities to the transition duct inlet boundary were $T_u = 5\%$ and $l = 0.07D_{h,s}$, where $D_{h,s} = 2\Delta R$. Supplementary simulations were completed to investigate the influence of the turbulence intensity over a range of constant values, and with fully-developed $k$ and $\varepsilon$ profiles (see App. E.3.4). Figure 4.2 plots the range of turbulence intensities based on a coarse 2D grid analysis without swirl. In particular, flow area blockage had a strong dependency on $T_u$ where the stronger turbulence resulted in thinner boundary layers. Similar magnitudes were observed on fine grids with the R$k$-$\varepsilon$ and SST turbulence models; however, the $C_{p_0}$ curve obtained from SST solutions had a negative slope with respect to $T_u$. Five percent turbulence intensity was ultimately selected for CFD simulations since it provided the closest agreement to experimental performance coefficients calculated for Configs. aa and bb (CB|OW) in unswirled flow (see Tab. E.12).

Turbulence length scale was investigated by Ng and Birk \cite{156} on a comparable geometry (S-Bend passages). A selected turbulence length scale of $l = 0.07D_h$ provided the best match to experiment. Order of magnitude lower and higher $l$ were also evaluated. No changes in pressure distributions were observed with the smaller value but the larger value, that was noted as being unphysical, resulted in pressure discrepancies at the first bend. Turbulence intensity was also varied
4.1. **COMPUTATIONAL DOMAIN**

![Graph showing turbulence intensity sensitivity comparison](image)

**Figure 4.2:** Configuration bb (CB|OW) turbulence intensity sensitivity comparison with respect to the $Tu = 5\%$ solution using the R$k$-$\varepsilon$ turbulence model on a coarse grid.

where $Tu = 10\%$ agreed better with experiment than the smaller tested values; however, $Tu = 4.2\%$ was measured [82].

### 4.1.2 Grid Generation

Given the number of different geometries evaluated, grid automation was essential. Executable scripts were created for both coarse and fine grids with user-defined control of the average axial grid space $dx$ and a coordinate input file that defined the CB and OW points $D–G$ and $S–U$. Appendix C includes sample script files for generating a coarse 2D grid in *ICEM CFD* and obtaining a converged solution in *Fluent*. Parameters relating to boundary layer growth and clustering of cells in regions with higher gradients were tuned to yield grids that minimized cell quantities but were of acceptable quality: aspect ratio below 30 (60 for 3D grids), volume change between adjacent cells below 2.0, and cell skewness above 0.5.

Figure 4.3 illustrates a 2D coarse mesh. All mesh variables were proportionate to the average...
axial grid spacing within the diffuser, $dx$, where the chosen coarse grid spacing was $dx/D_o = 0.011$ with an average first cell space from the wall of $dy_1 \approx 0.8dx$. Twenty-two cells spanned the inlet height $\Delta R_t$. Non-equilibrium wall functions were specified to resolve the boundary layers. The chosen 2D fine grid spacing was $dx/D_o = 0.0022$ with boundary layer refinement to achieve $dy_1 \approx 0.15dx$ that provided sufficient resolution to implement enhanced wall treatment for solving the boundary layers. Geometric multipliers increased the cell height away from the wall at a rate of 1.05–1.10.

![Figure 4.3: Config. bb (CB|OW) 2D coarse grid mesh. Red arrows define mesh driving directions. Fine grids were geometrically similar but with refined boundary layers.](image)

The transition duct and plenum did not possess regions with steep gradient changes, so cell density was coarsened away from the diffuser. For example, the first axial space in the transition duct was $2dx$ and the last axial space in the plenum was $4dx$. Steep gradients were expected around points $D$ and $U$, so axial spaces were reduced to $0.1dx$ and $0.14dx$ respectively. For 3D grids, the blocks originating from wall $FG$ were divided around the red arrow such that the blocks nearest the centreline were defined as central o-grid blocks. An outlet mesh used in the tab study on a $90^\circ$
4.1. COMPUTATIONAL DOMAIN

wedge is shown in Fig. 4.4.

Figure 4.4: Config. bc 3D coarse 90° wedge outlet mesh.

The challenge associated with implementing a totally structured grid required a method for growing the boundary layer off the centre-body to prevent the occurrence of a numerical shear layer downstream. Figure 4.3 shows how the mesh originating from the transition duct lifts off from the centre-body at point $F$ and wraps around to terminate at the plenum inlet. The compromise involved specifying the height of the first cell at the trailing edge of the centre-body as $2dy_1$. Grid designs were satisfactory with grid sensitivity evaluations provided in Apps. B.4, E.3.2 and F.2.2.

Plenum Sizing

Four plenums were modelled for Config. bb (CB|OW) and simulated with S0 and S20 swirl on coarse 2D grids. Plenum dimensions (length, height from centreline to plenum top, upstream length
from diffuser exit of radius $r_U$ to plenum inlet) were $(r_U, 1.5r_U, 0.04R_o), (3r_U, 2r_U, 0.12R_o), (8r_U, 4r_U, 0.20R_o)$, and $(20r_U, 11r_U, 0.39R_o)$. The larger plenums were closer to satisfying the requirements of no pressure gradients and zero velocity to correspond to the normal to boundary condition direction specification. With S0 swirl, difference in performance with respect to the chosen $(8r_U, 4r_U, 0.20R_o)$ plenum, the chosen plenum is shown in Fig. 4.1 was less than 0.4%. With S20 swirl, the plenum influence was more appreciable: the smallest two plenums had differences in performance that exceeded 10% whereas the largest plenum differences were below 0.1% with respect to the chosen plenum.

4.2 Solution Procedure

The governing equations were discretized with the QUICK scheme [157, 158] for the convective terms and the second-order scheme for pressure. The solution procedure coupled the pressure and velocity equations using the SIMPLE-C algorithm [159] with the skewness factor set to one owing to the presence of skewed cells near a conical centre-body.

The efforts to designing structured grids was beneficial for convergence; however, a methodology was developed for simulations without tabs to ensure stability during the iterative process:

1. The diffuser domain was initialized with average flow properties representative of the transition duct inlet profile and the plenum domain was patched with nearly quiescent atmospheric values.

2. Initial iterations were completed with constant density, wall functions and 1st order accurate discretization.

3. Fine grid simulations switched to enhanced wall treatment.

4. Ideal gas law and 2nd order accurate discretization were added.

5. Fine grid simulations switched turbulence models after converging the Rk-ε solution.
In all cases, when a change was enacted, the first 100 iterations were completed with reduced under-relaxation factors.

Given that the flow problem can be categorized as duct flow, so the initial mass-flow estimates in the diffuser satisfied continuity with respect to the transition duct inlet boundary condition, mass residuals were not included in the convergence criteria. Convergence criteria for the remaining global scaled momentum and turbulence residuals was $10^{-8}$ for coarse grids and $2 \times 10^{-6}$ for fine grids.

Simulations with tabs were first converged assuming the tab surface was an interior boundary condition. Performance was within 1% of the respective coarse 2D grid solutions that confirmed the 3D grids did not introduce excessive numerical diffusion. The tab surface was then defined as a wall boundary condition.

CFD was completed using four nodes on an *Intel i7* Quad core processor with 24 GB memory where eight computers were available for the task for multiple simulations. Computational expense was reasonable and accommodated the simulation of many different configurations: on the order of 11 minutes per 100 iterations for 1.4 million cells on fine 3D grids used in the optimization study and 6 minutes per 100 iterations for 500,000 cells on fine 2D grids.
Chapter 5

Manufactured Components

Three centre-bodies (CB) and four outer walls (OW) were manufactured. The CBs and three smaller OW geometries were selected from the results of the optimization study given in App. B based on their predicted performance capabilities:

(a) configuration aa: best solution with maximum outlet velocity uniformity and minimum total pressure loss

(b) configuration bb: best solution assuming equally weighted objectives (depicted in Fig. 1.1)

(c) configuration cc: best solution with maximum static pressure recovery

The configuration notation used throughout this thesis to define the diffuser parts is Config. CB|OW. The outer wall with \( AR = 6.18 \), OWd, was manufactured since literature recommended that augmentation devices were only beneficial in wide diffusers that experience stall.

Figure 5.1 shows the symmetry profiles of the manufactured CB and OW parts. Figure 5.2 is a photograph showing the CBb and OWb parts (the tufts were replaced with finer thread after the picture was taken). Engineering drawings for the manufactured parts used in this study are included in Appendix D. The measured values for the seven variables identified in Fig. 5.1 were from the actual components that were in close agreement to the values defined and provided in Fig. B.5.
Table 5.1 lists the values of the initial expansion angle, $\alpha_1$, that were obtained from interchanging the CBs and OWs.

Table 5.1: Manufactured components initial expansion angle, $\alpha_1$ (degrees). OWd was only tested with CBb.

<table>
<thead>
<tr>
<th>CB \ OW</th>
<th>a</th>
<th>b</th>
<th>c</th>
<th>d</th>
</tr>
</thead>
<tbody>
<tr>
<td>a</td>
<td>13.7</td>
<td>15.8</td>
<td>20.9</td>
<td></td>
</tr>
<tr>
<td>b</td>
<td>12.0</td>
<td>14.0</td>
<td>19.1</td>
<td>31.4</td>
</tr>
<tr>
<td>c</td>
<td>9.8</td>
<td>11.9</td>
<td>17.0</td>
<td></td>
</tr>
<tr>
<td>n</td>
<td>82.8</td>
<td>84.9</td>
<td>90</td>
<td></td>
</tr>
</tbody>
</table>

The outer walls were fabricated as two separate cones, $ST$ and $TU$, from aluminum sheetmetal with a continuous butt-weld seam on the exterior, and the interior was buffed to a smooth finish. Flanges were fillet-welded to the ends $S$ and $T$. The OW connection at point $T$ was also smoothed after being bolted together; however, the method of attachment to the transition duct did not give a perfectly smooth connection and lips could be felt. The outer walls OWa, OWb, and OWc had inlet diameter $D_o \equiv 2r_S = 6$ in, length $L = 1.5D_o = 12\Delta R_t$, diffuser $TU$ wall angle $\alpha_{TU} = 6^\circ$, and radial tolerance of $\pm 0.5$ mm. Table 5.2 lists the geometric features of the outer walls. The equivalent expansion angle, $\alpha_{SU} - \alpha_{DH}$, assumed a straight inner wall terminated on the centreline at the outlet, pt. $H = (x_U, 0)$. The conical diffuser equivalent angle, $2\theta_d$, was determined from $\tan \theta_d = (r_e - r_i)/L$ where $r_t = \sqrt{A_t/\pi}$.

Table 5.2: Manufactured outer wall parameters. Conical equivalent angle $2\theta_d$ determined from $\tan \theta_d = (r_e - r_i)/L$ where $r_t = \sqrt{A_t/\pi}$.

<table>
<thead>
<tr>
<th>OW</th>
<th>$2r_U$, mm</th>
<th>$AR$</th>
<th>equivalent expansion angle, $\alpha_{SU} - \alpha_{DH}$, $^\circ$</th>
<th>conical equivalent $2\theta_d$, $^\circ$</th>
</tr>
</thead>
<tbody>
<tr>
<td>a</td>
<td>130.3</td>
<td>1.61</td>
<td>11.0</td>
<td>6.9</td>
</tr>
<tr>
<td>b</td>
<td>140.4</td>
<td>1.91</td>
<td>12.5</td>
<td>9.7</td>
</tr>
<tr>
<td>c</td>
<td>168.0</td>
<td>2.73</td>
<td>15.7</td>
<td>16.3</td>
</tr>
<tr>
<td>d</td>
<td>252.6</td>
<td>6.18</td>
<td>26.3</td>
<td>36.6</td>
</tr>
</tbody>
</table>
Figure 5.1: Manufactured components geometry (symmetry plane shown). Configurations identified by (CB|OW).

Figure 5.2: Picture of centre-body, CBb, and outer wall, OWb, parts.
The centre-bodies CBa, CBb, and CBc were milled using single point incremental forming of aluminum sheets, and have lengths of 0.73, 0.76, and 0.84\(D_o\). CBn utilized the throat face. The method for manufacturing a CB resulted in its trailing edge (TE) being rounded by a radius of \(\sim 4\) mm and radial misalignment, with respect to the annulus’ inner cylinder, of \(\pm 1.1\) mm. The CBs had sleeve extensions that slipped on top of the throat end to maintain a smooth transition. Surface roughness was measured on the order of 2 \(\mu m\). Based on a Reynolds number of \(Re_t = 10^5\), the friction factor for 2 \(\mu m\) roughness is within 6% of a smooth pipe. A centre-body was bolted in place on its axis with a #8-32 nut located at the CB TE and was smoothed with plasticine (see Fig. 3.6). The CB TE at point G, shown in Fig. 5.1, were extrapolated.

The manufactured components had intentionally kinked CB corners, \(D–F\), and OW corners, \(S–T\). The OW flanges guaranteed crisp corners; however, slight curvatures, with axial lengths up to 0.05\(D_o\), existed at the CB points.

The annular diffuser systems with OWs a–c were sectioned into two separate components: a conical expansion section located above the CB and a solid fully open outer diameter diffusion section \(TU\). Area distributions and pressure recovery coefficients to an assumed streamline are shown in Fig. 5.3 for Configs. aa, bb, and cc. The streamline shown is from Config. bb. \(C_p\) was calculated from the actual area ratio whereas \(C_{p,eff}\) was calculated assuming boundary layer growth on the walls that reduced the effective inviscid area. Displacement thickness (Eq. (2.16)) was added to both surfaces for curvilinear wall distance \(x\). Effective inviscid area ratios increased to 1.64, 1.95, and 2.79 for Configs. aa–cc respectively.

The area distributions in Fig. 5.3 show that the region between \(DG\) and \(TU\) had decreasing area. This was an artifact of the evolutionary process because solutions with always increasing area distributions were possible. Static pressure loss occurred, so similar additional configurations were simulated that had positive \(AR\) growth throughout and confirmed that the selected designs were optimized. Schaefer, Hofman, and Gieß also reported that their optimized annular diffuser had a section that decreased in area.
5.0.1 Tabs

Three sets of tabs were manufactured from 22 gauge (0.64 mm-thick) aluminum sheet metal with nominal (width × height=#w×#h) and actual dimensions defined in Tab. 5.3 correlating to the parameters in Fig. 5.4. The listed values were taken as an average of four tabs of the same width (±0.04 mm) where height readings were within ±0.3 mm and angle measurements were within ±1° of the averages. Tabs were placed on the centre-body with double-sided tape at various locations such that the projected tab surface was normal to the rig x-axis.

The smaller tab height was defined based on recommendations [109] that it should be on the order of the boundary layer thickness and the most effective orientation angle was \( \phi = 135^\circ \). The square shape was manufactured for simplicity and in recognition that shape was not overly influential. The range of widths was selected to encompass a per-tab projected area blockage at the diffuser inlet of 1%. 

Figure 5.3: Area distributions and pressure coefficients for Configs. aa (dashed), bb (solid), and cc (dash-dot) based on the flow path of an assumed streamline.
### Table 5.3: Manufactured tab dimensions

<table>
<thead>
<tr>
<th>nommal width</th>
<th>actual width, $w$, mm</th>
<th>height, $h$, mm</th>
<th>tab angle $\phi$, $^\circ$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Nominal</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>5w</td>
<td>5.04</td>
<td>5.68</td>
<td>134.1</td>
</tr>
<tr>
<td>15w</td>
<td>15.70</td>
<td>6.12</td>
<td>132.0</td>
</tr>
<tr>
<td>25w</td>
<td>25.48</td>
<td>6.06</td>
<td>137.2</td>
</tr>
</tbody>
</table>

**Figure 5.4:** Square tab schematic for constant width, $w$.

The orientation shown in Fig. 5.4 has the 8h segment resting against the base whereas the tabs could be flipped with the 5h segment resting against the base and projecting outwards at a height of 8h. The axial position was denoted by the position of the projecting part of the tab’s bottom edge $x$-location and when multiple tabs were used, they were equispaced around the circumference. For example, Fig. 5.5 shows (location-number $\times$ width $\times$ height) $D4 \times 15w \times 5h$ tabs secured at the centre-body base.

**Figure 5.5:** Annular diffuser cutaway with tabs
Chapter 6

Results

Three studies were completed to individually assess the parameters experimentally and computationally:

6.1 Diffuser Geometry Study: evaluation of 12 configurations using the 4 CBs and 3 OWs without swirl.

6.2 Swirl Study: evaluation of 4 swirlers in \( AR = 1.91 \) diffusers using the 4 CBs.

6.3 Tab Study: evaluation of 4 OW diffusers with tabs and swirl.

6.1 Diffuser Geometry Study

Through interchanging the centre-bodies (CBs a–c and no centre-body, CBn) and outer walls (OWs a–c), 12 configurations were tested with S0 swirl and no tabs to evaluate the subtle geometry changes. Analysis of Configs. aa, bb, and cc (CB—OW) was sufficient to characterize the trends. Additional details of the diffuser geometry study with S0 swirl are in App. E.
6.1. DIFFUSER GEOMETRY STUDY

6.1.1 Flow Description

Figures 6.1(a)–(c) show CFD coarse 2D grid (-c) solutions of static pressure contours and streamlines on symmetry planes for Configs. aa, bb, and cc. A $p = 0$ contour was not included because pressure did not reach atmospheric until downstream $\sim 0.6D_o$ on the centreline into the wake, proving the need for an outlet plenum in numerical computations. It was clear that although Config. cc had the greatest pressure recovery due to the largest outlet area, it came at the cost of the largest recirculation zone at the end of the centre-body. For Config. nb, the streamlines provided in Fig. 6.1(d) followed flow patterns that were characteristic of a backward facing step: the sudden expansion produced a massive separated region that restricted pressure recovery.

Radial pressure gradients were evident in the pressure contours:

- Low pressure bubbles that were a result of convex curvature at point $D$, corresponded to a localized increase in velocity.

- Curvature effects that originated from point $T$ resulted in a higher static pressure region along panel $FG$.

In addition to the curvature effects, the flow, particularly along the outer walls OWa and OWb, experienced a Venturi effect whereby at point $T$ there was a local static pressure loss leading up to $x = D_o$ and sudden pressure recovery thereafter. From conservation of momentum, the bubble with decreased pressure induced acceleration upstream and deceleration downstream.

Figure 6.2(a) shows the predicted progression of axial velocity profiles within the diffusers. Conservation of axial momentum was apparent where the decrease of velocity of the high-momentum fluid correlated well with the rise in pressure observed in Figs. 6.1(a)–(c). The effect of radial curvature on the pressure contours was not as evident on the velocity profiles: for example, the curvature associated with point $D$ did not appreciably alter the diffuser inlet velocity profiles.

The downstream shape of the diffuser did not affect the inlet velocity profile; however, it was evident that the larger flow expansion angles, e.g. $\alpha_1$, resulted in worsening CB boundary layer
stability to coincide with the streamline separation points identified in Figs. 6.1(a)-(c) and larger magnitudes of $k$ closer to the centreline, as shown in Fig. 6.2(b).

### 6.1.2 Comparison to Experiment

#### Wall

Figures 6.3(a)-(d) show static wall pressure distributions for the selected configurations. The distributions were discontinuous at points $D$ and $T$ due to the modelled sharp corners. The ideal pressure
distribution was calculated from normal areas to an assumed streamline. Standard and RNG distributions were not shown but similar to the realizable $k$-$\varepsilon$ fine grid (-f) solutions where Rk-$\varepsilon$ estimated the nominally highest diffuser inlet pressure and std $k$-$\varepsilon$ the lowest. 

The CFD static pressures were non-dimensionalized by the experimental dynamic pressure to isolate the source causing the deviations in $p_t$. For Configs. aa and bb, excellent comparison occurred for $x > 0.2D_o$; however, the SST solutions predicted closer pressure at section $\odot$. All turbulence models satisfactorily predicted the slight dip in static pressure immediately downstream of
Figure 6.3: Diffuser geometry study wall pressure distributions Configs. aa and bb.
Figure 6.3: Diffuser geometry study wall pressure distributions Configs. cc and nb. CB data (solid symbols) on left axis and OW data (hollow symbols) on right axis. Error bars with symbols denote pressure readings taken by the 3-hole probes nearest to the respective wall. CFD non-dimensionalized by the experimental $\langle q_{t,exp} \rangle$. 
CHAPTER 6. RESULTS

For Config. cc, Fig. 6.3(c) shows that the $k$-$\varepsilon$ models deviated from experiment below $x = 0.6D_o$ whereas the SST model captured the proper magnitude beyond $x = 0.2D_o$. The apparent better-than-ideal performance of the $k$-$\varepsilon$ models was due to the over-estimation in mass flow rate that gave a higher dynamic pressure than measured in experiment.

The predicted Config. nb OW distributions shown in Fig. 6.3(d) were similar to experiment. The trends from the other eight configurations were similar to their respective OWs in that the shapes were in agreement but there were discrepancies in magnitudes—particularly at the inlet.

Outlet

Figure 6.4 shows outlet axial velocity profiles for Configs. nb, aa, bb, and cc. The shown experimental profiles were azimuthally-averaged with the error bars depicting one standard deviation at the given radial location. Velocity profiles for the remaining configurations are shown in Fig. E.19. Good agreement between CFD and experiment occurred for Configs. aa and bb. For Config. cc, the experiment obtained greater core flow recovery than that predicted by CFD.

![Diffuser geometry study outlet axial velocity profiles.](image)

Figure 6.4: Diffuser geometry study outlet axial velocity profiles.
6.1.3 Performance Summary

Table 6.1 lists the performance parameters for Configs. aa, bb, and cc. Figures E.20(a) and (b) plot the performance of all tested configurations. Pressure recovery increased with area ratio for the well-designed $L/\Delta R_t = 12$, $AR \leq 2.73$ diffusers but outlet velocity uniformity was highest in the smallest $AR = 1.61$ diffusers (OWa). CFD with the different simulated turbulence models generally predicted similar outlet velocity profiles and wall pressure distributions: virtually no difference was observed in the $k-\varepsilon$ solution performance parameters that were better at predicting $\gamma$ whereas the SST turbulence model predicted closer $C_b$ to experiment.

<table>
<thead>
<tr>
<th>Table 6.1: Diffuser geometry study performance.</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
</tbody>
</table>
lower \( k \) nearer to the centreline. These profiles were representative of the other swirl angles that failed to identify any appreciable differences due to the CB shape. Forthcoming analysis was based on the CBB simulations but were valid for CBA and CBB.

Figures 6.6(a) and (b) show the predicted static pressure contours and streamlines for Configs. bb and nb with S20 swirl. In comparison to Fig. 6.1(b) that shows predicted pressure contours without swirl, Fig. 6.6(a) shows more severe radial pressure gradients, flow separation was delayed, and reattachment occurred further downstream. For Config. nb, with reference to the no swirl case in Fig. 6.1(d), the swirl caused more pronounced radial curvature effects associated with point \( T \) on pressure. It was not clear how the CTRZ was affected since the \( Rk-\epsilon \) and RNG models predicted longer recirculation regions with S20 whereas the std \( k-\epsilon \) and SST models predicted shorter regions than the S0 solution.

Figure 6.7 shows the influence of inlet swirl on the Config. bb profiles. The \( u \) and \( Tu \) profiles were quite similar in the conical expansion section and the S10 and SCV profiles were similar to the S0 solution in the solid diffusion \( TU \) section. Swirl angle magnitudes reflected the swirler used. The flow transport with the SCV swirler maintained the solid body rotation behaviour up to \( x = x_E \); further downstream, the SCV profiles were similar to the S10 profiles.

The S0, S10, and SCV pressure profiles were mostly similar with the exception just downstream of the CB TE where some deviation was observed in the S10 profiles. For the shown \( Rk-\epsilon \) solutions, the S0 and SCV separation region lengths were similar whereas shorter lengths were predicted with S10 swirl. Alternatively, SST predicted the shortest length with SCV.

With S20 and S40 swirl, the centrifugal forces created by the stronger swirl were large enough to induce sizeable CTRZ downstream of the CB TE. In addition to reversed flow in this separated region, free vortex behaviour was predicted along the centreline and there was a dramatic increase in \( k \). For S40 swirl, the expansion of the CTRZ towards the outlet formed a sizeable free shear layer that was marked by the noticeable peak in \( k \).
6.2. SWIRL STUDY

Figure 6.5: Swirl study predicted influence of CB shape on profiles with S20 swirl. R$k$-$\varepsilon$-f solution.
6.2.2 Comparison to Experiment

Wall

Figures 6.8(a)–(e) show the influence of swirl on pressure distributions for Config. bb. Figure 6.8(f) confirms that, similar to experiment, the influence of the different shaped CBs was small but not including a CB caused a different effect. For \( \leq S20 \) swirl, the standard and RNG solutions were similar to the \( Rk-\varepsilon \) distributions whereas with S40 swirl, Figs. 6.8(c) and (d) show that the \( Rk-\varepsilon \)
Figure 6.7: Swirl study predicted influence of inlet swirl angle on profiles. Config. bb Rk-ε-f solutions (1).
and RNG solutions were similar but the standard \( k-\varepsilon \) solution was more comparable to the SST distributions.

Up to and including S20 swirl, the simulations were capable of correctly predicting the shapes of the distributions where the SST solution was closest to experiment. For S40 swirl, there was reasonable agreement in the conical expansion section but deviations occurred in the wall \( TU \) distributions. The R\( k-\varepsilon \) solutions obtained closer agreement to the upstream solution than predicted by SST. Discussion of the decrease in pressure near the centre-body trailing edge is in App. F.1.

**Outlet**

Figures 6.9 and 6.10 compare outlet axial velocity and swirl angle profiles between experiment and CFD for the given amount of swirl. CFD profiles were in good agreement with experiment for Config. bb with S10, S20 and SCV swirl where the \( k-\varepsilon \) models predicted closer \( u_e(r) \) but the SST solution was a better match to \( \vartheta_e(r) \). With S40 swirl, the R\( k-\varepsilon \) model predicted the correct magnitude of the high momentum region near the outer wall but over-estimated its size. Similarly to the CBn analysis with S0 swirl, the predicted velocity profiles with S20 were more peaked than
Figure 6.8: Swirl study wall pressure distributions. Config. bb with S10 and S20 swirl.
Figure 6.8: Swirl study wall pressure distributions. Config. bb with S40 swirl.
Figure 6.8: Swirl study wall pressure distributions Config. bb with SCV swirl and all CB Configs. with OWb. CB data (solid symbols) on left axis and OW data (hollow symbols) on right axis. CFD non-dimensionalized by the experimental $\langle q_t, exp \rangle$. 

(e) SCV 

(f) S20 Rk-$\epsilon$-f different CBs
actual; however, the SST model obtained a comparable $\theta_e(r)$ profile.

6.2.3 Performance Summary

Performance did not significantly depreciate for swirling inlet flow with swirl number $S_t \leq 0.7$ (S20 swirl) but the more aerodynamic curved-vane swirler with an average $10^\circ$ angle delivered the best performance and out-performed its straight-vaned equivalent by 2–5% in $C_b$ and 1% in $\gamma$. 
6.3. TAB STUDY

Appreciable performance improvements did not occur due to minor centre-body curvature changes that provided initial flow angles of \( 12^\circ < \alpha_1 < 16^\circ \). Table 6.2 lists the performance values for Config. bb in swirled flow.

<table>
<thead>
<tr>
<th>Table 6.2: Swirl study Config. bb performance.</th>
</tr>
</thead>
<tbody>
<tr>
<td>S10</td>
</tr>
<tr>
<td>C_b</td>
</tr>
</tbody>
</table>
| \begin{tabular}{l|lll|lll|lll} 
experiment & 0.58 & 0.94 & -0.096 & 0.58 & 0.91 & -0.081 & 0.19 & 0.71 & -0.280 \\
Rk-\varepsilon-c & 0.66 & 0.93 & -0.055 & 0.63 & 0.91 & -0.065 & 0.29 & 0.70 & -0.210 \\
Rk-\varepsilon-f & 0.65 & 0.93 & -0.060 & 0.63 & 0.91 & -0.071 & 0.30 & 0.72 & -0.208 \\
SST & 0.59 & 0.90 & -0.089 & 0.61 & 0.91 & -0.077 & 0.40 & 0.78 & -0.155 \\
\end{tabular} |

For \( \leq S20 \) swirl, the different turbulence models predicted similar solutions. When compared to experiment, the Rk-\varepsilon and RNG models were better at predicting the outlet axial velocity profile whereas the SST model predicted closer wall pressure distributions. At S40 swirl, the Rk-\varepsilon and RNG models—which included modifications to account for swirl—predicted more representative solutions.

6.3 Tab Study

Computational studies were completed on coarse 3D grids using the realizable \( k-\varepsilon \) turbulence model and wall functions with \( 15w \times 5h \) tabs placed at the centre-body base in configurations with CBb and OWs a–d: 4 and 6 tabs at S0 swirl; 3, 4, 6, and 8 tabs at S20 swirl, and 4 tabs at S10 and S40 swirl. Computational domains were modelled as wedges centred about a tab for assumed symmetry with periodic boundary conditions assigned to the meridional surfaces. Experimentation with tabs was completed at S0 and S20 swirl.

Evaluation of Config. bd without tabs is in App. G.
6.3.1 Flow Description

Figure 6.11 shows predicted pressure fields and streamlines through the Config. bc diffuser with S20 swirl on a modelled 90° wedge for assumed symmetry with (location-number×width×height) D4×15w×5h tabs. The analysis with S0 swirl, that predicted similar features, is in App. H.1. In comparison to Fig. H.1 that shows contours of a simulation with S0 swirl, the recirculation region downstream of the tab with S20 swirl was a well-defined oval shape and the major axis was angled at $\tan^{-1}(\theta_{rn}/2a) \approx \vartheta_t = 11^\circ$.

\[ p\langle q \rangle: -1.2 -1 -0.8 -0.6 -0.4 -0.2 0 \]

\[ x/D_o = 0.23 \]

\[ \theta=0^\circ \text{ plane} \]

\[ \theta=45^\circ \]

\[ \text{Figures 6.12-6.16 plot axial velocity, total pressure, axial and azimuthal vorticity, and turbulence intensity distributions on axial planes with added normal velocity vectors and streamlines. The contours confirm symmetry about the separated region major axis. Noteworthy features that were similar at the other simulated swirl angles include:} \]
• A tab-induced static pressure hill at the diffuser inlet developed strong local radial vorticity on streamlines that forced flow outwards due to the pressure hill along the inner wall, which increased velocity and reduced pressure as $|\theta - 45^\circ| \to 0^\circ$.

• The near-wake of tab was a dead zone with negative total pressure that grew in size but weakened in strength further downstream.

• High axial vorticity was caused by the flow detouring around the tab. The recirculation zone eccentricity was $e \approx 0.80$–$0.87$ (Config. bd predicted $e \approx 0.71$) with major axis length $2a/D_o = 0.04AR + 0.29$ (Config. bc did not follow this trend where $a = 0.04D_o$).

• Outward spreading hairpin structures rode on top of the CVP and $\xi_a$ was originally at least 4 times greater than $\xi_x$. For Config. bc with S20 swirl, fifty percent of the azimuthal vorticity strength vanished by $x \approx 4h = 0.16D_o$.

• Turbulence kinetic energy contours were similar in shape to the $\xi_a$ contours and maximum intensity occurred at $x \approx 0.39D_o$.

The difference of a swirling flow with respect to no swirl was the rotational aspect of the contours. Based on total pressure contours shown in Fig. 6.13, the tab indentation (defined by the negative total pressure region on the $x = 0.13D_o$ plane) for Config. bc with 4 tabs rotated $75^\circ$ CCW upon reaching the outlet. The rotation affected the strength and dissipation of the CVP. Counterclockwise structures were approximately double in original strength with respect to the CW structures but lost 50% of the original strength at $x = 8h = 0.33D_o$. Clockwise structures retained 50% of the original strength until $x = 12h = 0.46D_o$; however, Fig. 6.14 shows that the CCW structure was stronger at the $x = 0.52D_o$ plane, so the 50% threshold for the CW structure was within the tolerance of considering the flow irrotational. The CVP for the other configurations with four tabs vanished earlier.
Figure 6.12: Config. bc with D4 $\times$ 15w $\times$ 5h tabs and S20 swirl axial velocity contours and velocity vectors.

Figure 6.13: Config. bc with D4 $\times$ 15w $\times$ 5h tabs and S20 swirl predicted total pressure contours.
Figure 6.14: Config. bc with D4×15w×5h tabs and S20 swirl predicted axial vorticity contours.

Figure 6.15: Config. bc with D4×15w×5h tabs and S20 swirl predicted azimuthal vorticity contours.
6.3.2 Tab Wake

Figures 6.17, 6.18, and 6.19 show predicted surface pressure and the extents of the recirculation regions for Configs. bb, bc, and bd with 4 tabs and S20 swirl in the tab wake. The quadric surface was created $\delta = 5$ mm above and parallel to panel $DE$:

$$x^2 + 8.0486y^2 + 8.0486z^2 + 3.8855xyz - yz + 3.8855xz = 8.5611r_Q^2$$  \hspace{1cm} (6.1)

where $r_Q = (r_D + \delta)\cos|\alpha_{DE}|$.

The CVP in the CFD solutions of Configs. bc and bd had stable foci and the recirculation region in Config. bd was fed from reversed outlet flow entrainment. The wake in Config. bb, that was similar for Config. ba, had noticeable asymmetry. The Reynolds number based on the tab width for the four outer wall configurations was $4 - 5 \times 10^4$. Periodic shedding was found to occur behind a circular cylinder over the range $10^2 < Re_D < 10^7$ with an average Strouhal number of
Since alternating vortices were likely shed from the tabs based on the tab Reynolds number falling within the range when shedding occurs, simulating unsteady-RANS and ensemble averaging one period would give a more representative time-averaged solution. Further investigation was necessary to evaluate if the obtained steady-state solutions had any merit in representing the
Figure 6.19: Config. bd with D4×15w×5h tabs and S20 swirl predicted quadric surface pressure and streamlines.

tab wake flow physics. Stable wake solutions, as shown in Figs. 6.18 and 6.19 were expected. Coincidentally, Strouhal numbers of 0.17–0.20 were measured in the unstable-wake CFD solutions.

From potential flow theory, the profile of the recirculation region resembled flow past a cylinder that was defined by a uniform stream plus a source-sink pair. The stream function was expressed as:

$$\psi = V \sin \theta \left[ r - \frac{b^2}{r} \right]$$  \hspace{1cm} (6.2)

Since the shape was ovular, its radius was defined by:

$$r^2 = \left[ (x-x_0) \frac{b}{a} \right]^2 + (z-z_0)^2$$  \hspace{1cm} (6.3)

and angle by:

$$\tan \theta = \frac{z-z_0}{x-x_0}$$  \hspace{1cm} (6.4)

where the axial centre of the oval was \((x_0, z_0)\) and for convenience of representation in 2D, \(z \equiv r\theta\)
with \( z_0 = 0 \). The semi-major axis, \( a \), was defined from:

\[
a^2 = \frac{(1-e^2)(-x_0^2) + \left(\frac{w}{2}\right)^2}{1-e^2}
\]  

(6.5)

for a tab width, \( w \), and semi-minor axis, \( b \), as:

\[
b = a\sqrt{1-e^2}
\]  

(6.6)

Figure 6.20 plots the stream function \( \psi \) assuming \( e = 0.83 \) and \( x_0 = 0.18D_o \). Even though the constants were selected to guarantee a good match to Fig. H.8, it was evident that potential flow theory can be used to mimic the geometrical effects of a tab to validate the incentive of developing a vortex generator model for the tabs. (Recall that VG models were not intended to model the flow physics.)

Figure 6.20: Tab wake potential flow solution.

Given that the recirculation region major axes were basically straight and eccentricities were similar to those calculated with S0 swirl (see App. H.1.3), a developed vortex generator model for tabs should include swirl angle—i.e. for the 2D equation, \( \psi = f(V,w,e,x_0,z_0,\vartheta) \)—in order to obtain a characteristic shape that was representative for the flow properties evaluated in this investigation.
6.3.3 Number of Tabs Experimental Study

This study was completed using Config. bb with S20 swirl. Figure [6.21] shows outlet total pressure profiles with in-plane velocity vectors nondimensionalized by $\langle V_e \rangle$ using the given number of $15w \times 5h$ tabs placed at point $D$ (four tabs depicted in Fig. [5.5]). The contours show the corresponding number of indentations that were rotated approximately $85^\circ$ CCW from the instigating tab at the diffuser inlet with lagging higher velocity distortions. The fluid particle degree of rotation was confirmed assuming the average of the inlet and outlet axial velocity values to obtain a particle residence time within the diffuser and a circumferential distance from the average tangential velocity that occurred at a constant radius of $0.7r_e$. Data were blanked in the centre for 0–2 tabs as a consequence of the flow angle exceeding the 7-hole probe calibration range (i.e. the flow was reversed) and corresponded to $u_{cl} = 0$ in Fig. [H.18(b)]

The contour plots show that two tabs were able to sufficiently reduce/eliminate the CTRZ beyond the outlet. The effective radius of blanked data descended from $0.18r_e$ with no tabs to $0.15r_e$ with one tab and $0.04r_e$ with two tabs. As the number of tabs was increased, the outlet axial and tangential velocity profiles, shown in Fig. [6.22(a)] and [b] had higher centreline velocity and more energy closer to the outer wall. The velocity profiles were azimuthally-averaged with error bars depicting one standard deviation at the given radial location (magnitude indicated was typical for all cases tested). Tangential velocity profiles, shown in Fig. [6.22(b)], confirmed that the flow was dominated by solid body rotation in the core $r < 0.5r_e$ (see Eq. (F.1) where $C_1 \gg C_2$ with S20 swirl). The average ratio of outlet tangential to axial velocity with S20 swirl was 50%.

As the flow convected within the diffuser, higher axial velocity occurred closer to the outer wall since less outward diffusion was expected. Although the tabs were placed on the CB, the vortices generated by the tabs affected this high momentum region. The tab-induced vortices still had sufficient strength at the outlet so the indentations, as seen in Figs. [6.21(b)]–[g] were the primary reason for higher standard deviations in velocity nearer to the outer wall.
Figure 6.21: Influence of number of 15w×5h tabs at CB base on Config. bb outlet experimental total pressure contours and velocity vectors with S20 swirl.
By increasing the number of tabs, there was an increase in tab blockage area and linear proportional loss in $C_{p0}$, shown in Fig. 6.18(a), of $3.1A_{tab}$ from the flow distortion. In comparison, the tabs improved at least $0.6A_{tab}$ in $C_q$ as the CTRZ was reduced that was evident from larger centreline velocities. There was a resulting loss in $C_h$ of $2.6A_{tab}$ that was attributed to the fact that the diffuser was too well-designed, even with a strong swirling inlet flow, for augmentation devices to be implemented for the purposes of recovering pressure.

The velocity profiles in Fig. 6.22(a) became noticeably more inform from 0 to 6 tabs: axial centreline velocity increased and the difference between maximum and minimum velocity decreased. The 6- and 8-tab velocity profiles were similar enough to suggest that there may be an upper limit to $\gamma$ with this tab design. Six tabs recovered more $C_q$ than 8 tabs, which suggested that $A_{tab} \approx 0.07$ was preferential for the given amount of flow blockage $A_{Bf}$. Outlet velocity uniformity coefficients are given in Fig. 6.18(b).

Experiments that tested tab axial location and tab $w \times h$ shape were evaluated in App. H.2. The
6.3. TAB STUDY

analysis determined that:

- Tabs located at the centre-body base, in comparison to on the CB, caused the greatest amount of flow distortion that resulted in the most improvement to outlet velocity uniformity.

- Results were more strongly correlated to the tab blockage area ratio, $A_{tab}$, as opposed to the actual tab width and height. The five millimetre high tabs were successful at generating distortions since the inner wall boundary layer height of $\delta_i \approx 5$ mm was measured at section 1.

Appendix H.3 provides an experimental analysis of different numbers of tabs placed at the CB base in the different AR diffusers with S20 swirl. Similarly to the results described for Config. bb, $\gamma$ improved but penalties in back pressure were observed in Configs. ba and bc when tabs were added. Improvements to $C_b$ occurred in Config. bd with tabs; however, its performance was significantly worse that that of Config. bc.

6.3.4 Number of Tabs CFD Study

This study was based on simulations of 3, 4, 6, and 8 tabs located at the centre-body base in Configs. ba, bb, bc, and bd with S20 swirl. The predicted recirculation regions through using 3, 6, or 8 tabs were similar to their respective shapes shown in Figs. 6.17–6.19 with 4 tabs and S20 swirl. All predicted Config. bc wakes with S20 swirl were well-defined and angled at $\theta_t \approx 11^\circ$. Major axis lengths excluding Config. bc are given in Tab. 6.3 and were determined from the location where streamlines left and regained motion on the major axis. For example, Fig. 6.20 shows that stagnation points occurred at $x \approx -0.03D_o$ and $x \approx 0.39D_o$. Given the convergence issues, it was possible that $a$ was independent with respect to the number of tabs. No discernable trends were identified for eccentricity that was on average $e = 0.81$. This suggested that the distance between adjacent tabs had minimal influence on the shaping of the recirculation region. With 8 tabs, the distance between adjacent tabs was $0.3D_o$. 
Table 6.3: Predicted influence of area ratio on recirculation region length with S20 swirl and tabs.

<table>
<thead>
<tr>
<th># of tabs</th>
<th>major axis length</th>
</tr>
</thead>
<tbody>
<tr>
<td>3</td>
<td>$2a/D_o = 0.06AR + 0.21$</td>
</tr>
<tr>
<td>4</td>
<td>$2a/D_o = 0.04AR + 0.29$</td>
</tr>
<tr>
<td>6</td>
<td>$2a/D_o = 0.04AR + 0.21$</td>
</tr>
<tr>
<td>8</td>
<td>$2a/D_o = 0.02AR + 0.26$</td>
</tr>
</tbody>
</table>

6.3.5 Amount of Swirl

Figures 6.23(a)–(c) compare predicted performance from CFD simulations against swirl number. For Configs. ba–bc, performance was more strongly dependent on swirl than the vorticity generated by the tabs since the slopes predicted without tabs were similar to those with 4 tabs. For Config. bc, an unstable recirculation region with S0 swirl was predicted (see Fig. H.8) whereas S10 and S20 simulations predicted well-defined wakes (see Fig. 6.18); all simulations with Configs. ba and bb predicted noticeably asymmetric wakes. As swirl increased, the tabs became less useful in Configs. ba–bc at exciting the flow since the difference in $\gamma$ with respect to the 0-tab cases dropped at a rate of at least $-0.02S_t$ (excluding Config. bc with S0 swirl). Better mixing occurred when the tabs were normal to the swirl angle. For Config. bd, the difference in $\gamma$ increased at a rate of 0.34$S_t$.

The back pressure penalty associated with the tabs at higher swirl angles was predominantly a function of the diffuser area ratio. With S40 swirl and 4 tabs, the difference in $C_b$ with respect to the corresponding 0-tab case increased at a rate of $\exp(-2.8AR + 1.1)$. For Config. bd, the penalty associated from 4 tabs in $C_b$ with S40 swirl was below the penalty with S0 swirl. This was not an useful observation given the dramatic difference in flow profiles between no tabs that predicted a symmetric outlet profile biased towards the outer wall (Fig. G.10) whereas with 4 tabs there was a significant amount of reversed flow entrainment at the outlet (Fig. 6.28).

Although the CFD did not predict pressure recovery improvements for any of the simulated configurations with tabs, there was a clear dependency on the diffuser area ratio. Given $L/\Delta R_t = 12$, evaluating annular diffusers with $3 \leq AR \leq 6$ may establish a range for which adding vortex
Following the reduced ability to improve $\gamma$ at higher swirl angles, $\theta$, there was a drop in the $C_{p_0}$ penalty—this confirmed that the tabs were more streamlined. Configuration bc with S20 swirl was an exception that calculated a $C_{p_0}$ with 4 tabs 300% higher than without tabs. A convergence study was necessary to confirm this feature.

Tab wake recirculation region eccentricities with S10 swirl were measured on quadric surfaces of $e \approx 0.83$ and were angled in agreement with the swirl angle $\theta_t \approx 5^\circ$. With S40 swirl, flow reattachment after the tab occurred at $x < 0.03D_o$ for the tangentially dominated flow, $\theta_r \approx 26^\circ$, that prevented the possibility of estimating $e$. A CCW axial vortex structure was predicted that was confined to the tab corner.
6.3.6 Comparison to Experiment

Inlet

Figures 6.24(a)–(c) show inlet pressure profiles for Configs. bb, bc, and bd with S20 swirl and D4×15w×5h tabs. The experimental traverses were taken on the $\theta = 0^\circ$ axis (circumferentially offset from the tabs placed at $\theta = 45^\circ$) and two linear extractions were taken from the CFD data at $\theta = 0^\circ$ and $\theta = 45^\circ$. Total pressure matched between CFD and experiment. The CFD predicted higher static pressure inline with the tab. The prediction of static back pressure was different from experiment for each configuration: Config. bb over-predicted recovery, bc under-predicted, and bd matched.

Figure 6.24: Configurations with D4×15w×5h tabs and S20 swirl inlet pressure profiles. Normalized $p_0$ data refer to bottom axis and $p$ data to top axis. Solid symbols = CB distributions and hollow symbols = OW distributions. CFD non-dimensionalized by the experimental $\langle q_t, \text{exp} \rangle$. CFD results extracted along $\theta = 0^\circ$ and $\theta = 45^\circ$ planes. Experimental traverses were completed at $\theta = 0^\circ$. 
Wall

Figures 6.25(a)–(d) plot wall pressure distributions for Configs. ba–bd with D4×15w×5h tabs and S20 swirl. Centre-body taps were placed on the θ = 0° plane whereas OW taps were inline with the tabs at θ = 45°. The CFD OW distributions for Configs. ba–bc agreed well in the diffuser that correctly predicted the dip at x ≈ 0.2D₀. The predicted CB distributions showed the correct gradient near the trailing edge; however, a dip around x = 0.4D₀ was not observed in experiment.

For Config. bd simulations, the pressure dips were not as pronounced and the experimental OW distribution did not show the dips. The magnitudes, however, were in good agreement through the entire diffuser.

Outlet

Figures 6.26 shows a comparison of outlet p₀ and ξₓ contours between CFD and experiment for Config. bb with S20 swirl and D4×15w×5h tabs. The contours were well-predicted for Config. bb—both in shape and magnitude. CFD correctly predicted that the indentation had rotated approximately 90° with the lagging remnants of the CCW axial vortex. The core region had negative total pressure and high CCW axial vorticity due to the swirling motion.

Figure 6.27 shows outlet distributions for Config. bc. Magnitudes were well-predicted whereas CFD under-estimated the amount of rotation by roughly 20°. For Config. bd, Fig. 6.28 shows that CFD gave a reasonable representation of the total pressure contours. CFD predicted a flow rotation of 45° that agreed well with experiment. Flow visualization confirmed flow separation in the core region with highly fluctuating flow nearer to the outer wall. The experimental axial vorticity peaks for r < 0.5rₑ were erroneous. Although the circled CW and CCW vorticity structures occurred in the same areas, the experimental pair was likely not physical since it was in a predicted reversed flow region.
Figure 6.25: Configs. ba and bb with $D4 \times 15w \times 5h$ tabs and S20 swirl wall pressure distributions.
Figure 6.25: Configs. bc and bd with D4×15w×5h tabs and S20 swirl wall pressure distributions. CB data (solid symbols) on left axis and OW data (hollow symbols) on right axis. CFD non-dimensionalized by the experimental \( \langle q_{i,\exp} \rangle \). CFD results extracted along \( \theta = 0^\circ \) and \( \theta = 45^\circ \) planes. Experimental CB taps located at \( \theta = 0^\circ \) and OW taps at \( \theta = 45^\circ \).
Figure 6.26: Config. bb with $D4 \times 15w \times 5h$ tabs and S20 swirl outlet contours. CFD results plotted on top two quadrants and experiment on bottom two quadrants. Total pressure contours on left quadrants and axial vorticity contours on right quadrants.

Figure 6.27: Config. bc with $D4 \times 15w \times 5h$ tabs and S20 swirl outlet contours. See Fig. 6.26 for quadrant identifiers.
6.3. TAB STUDY

Figure 6.28: Config. bd with D4×15w×5h tabs and S20 swirl outlet contours. See Fig. 6.26 for quadrant identifiers.

Performance

Figures 6.29(a)–(c) plot performance versus tab blockage with S20 swirl and none, 3, 4, 6, and 8 equally spaced 15w×5h tabs at the CBb base. Experimental Config. bc and bd outlet mass-flow averaged quantities were evaluated using \( \dot{m} \) (see Fig. 6.27 for \( C_{p_{0}} \) coefficients using \( \dot{m} \)). The recalculation in mass-flow averaged quantities occurred, particularly with OWd, since \( \dot{m}_{e} \) was 30–50% greater than \( \dot{m}_{t} \) as a consequence of conversion executable for the 7-hole probe raw data not including an option that described the observed reversed flow situation. For Config. bd, \( \gamma \) was on average 40% lower and \( C_{p_{0}} \) improved by < 10% using \( \dot{m}_{t} \).

Figure 2.5 shows that first appreciable stall of \( L/\Delta R_{t} = 12 \) diffusers occurs at \( AR \approx 2.78 \). Table 6.4 shows adjusted area ratios to account for the 15w×5h tabs that block 0.012\( A_{f} \) at the centre-body base. Further investigation with Config. bc would be beneficial to determine the coupled influence of periodic shedding from the tabs and the suspicion of appreciable stall due to the increased area ratio from the tab blockage.
Figure 6.29: Tabs study performance with S20 swirl. Solid lines = exp, dashed lines = CFD.

Table 6.4: Adjusted area ratios due to tab blockage from 15w×5h tabs placed at the CB base.

<table>
<thead>
<tr>
<th></th>
<th># of tabs</th>
</tr>
</thead>
<tbody>
<tr>
<td>OW</td>
<td>0</td>
</tr>
<tr>
<td>a</td>
<td>1.61</td>
</tr>
<tr>
<td>b</td>
<td>1.91</td>
</tr>
<tr>
<td>c</td>
<td>2.73</td>
</tr>
<tr>
<td>d</td>
<td>6.18</td>
</tr>
</tbody>
</table>

Figure 6.29(d) quantifies the amount of flow inlet area blockage. Experimental values were lower and consistent with the 0-tab $A_{B_1}$ due to the 3-hole probes being out of phase with the tabs whereas the CFD was determined by integrating over the entire cross section. CFD predicted the largest amount of aerodynamic blockage with 3 tabs. Higher blockage associated with Configs. bc
and bd may be due to the prediction of stable recirculation regions in the tab wake that, as shown in Figs. 6.17–6.19, produced stronger pressure hills upstream of the tab.

Overall, performance of Config. bd with S20 swirl and tabs was well-predicted. Figures 6.24(c) and 6.28 show matching inlet and outlet profiles with experiment and Fig. 6.25(d) shows matching wall pressure distributions with 4 tabs. The agreement was due to a cancellation of errors since the pressure distributions in Figure G.7 show that the back pressure associated with the diffusion and no swirl was over-predicted. Adding S20 swirl as seen in Fig. G.9 did not introduce additional errors. The agreement seen in Fig. 6.25(d) with tabs therefore must under-predict the pressure penalty caused by the tabs. It just so happens that the Config. bd prediction with 4 tabs provided an equivalent penalty to achieve the same $C_b$ as experiment, as shown in Fig. 6.29(a).

Performance magnitudes and slopes with respect to tab blockage for simulations with tabs were generally in good agreement but better $C_b$ was not predicted with tabs over the respective no-tab configuration. This was evident in the Config. bd results where the $C_b$ with 0 tabs was 47% over-predicted. It may be beneficial to repeat the analysis with the SST model that predicts more aggressive boundary layers in diffusing flows.

Table 6.5 quantifies the amount of tab blockage for experiment and CFD that obtained the highest γ. Although the data sets were not the same, the trends confirm that a correlation exists between area ratio and the preferable amount of tab blockage.

Table 6.5: Number of tabs to obtain maximum outlet velocity uniformity.

<table>
<thead>
<tr>
<th></th>
<th>OW</th>
<th>a</th>
<th>b</th>
<th>c</th>
<th>d</th>
</tr>
</thead>
<tbody>
<tr>
<td>experiment</td>
<td>4</td>
<td>6</td>
<td>2</td>
<td>6</td>
<td></td>
</tr>
<tr>
<td>$Rk-\varepsilon-c$</td>
<td>6</td>
<td>3</td>
<td>4</td>
<td>8</td>
<td></td>
</tr>
</tbody>
</table>
Chapter 7

Conclusions and Recommendations

This chapter acknowledges that a research project of this scope carries a wealth of information beyond the diffuser assessment:

7.1 Conclusions: key results that satisfy the objective.

7.2 Design Advice: a sharing of the experience acquired through a dedicated focus on annular diffusers.

7.3 Benefits for Industrial Sponsor: suggestions for how to profit from this applied research investigation.

7.1 Conclusions

- The numerical optimization methodology was successful at identifying short annular diffuser geometries of length $L/\Delta R_i = 12$ with better performance. Experimentation of the manufactured geometries substantiated the relationships that pressure recovery improved but outlet velocity uniformity decreased as area ratio increased for $1.61 < AR < 2.73$. 
7.1. CONCLUSIONS

- The twisted-vaned $10^\circ$-equivalent inlet swirler delivered the best back pressure and lowest outlet distortion in comparison to straight-vaned swirlers with angles $\vartheta = 0^\circ$, $10^\circ$, $20^\circ$ and $40^\circ$. Performance did not significantly depreciate for $\leq 20^\circ$ inlet swirl.

- Square tabs with height on the order of the boundary layer thickness improved the outlet velocity uniformity by reducing the size of the central toroidal recirculation zone created by a strong swirling inlet flow but incurred back pressure penalties at all tested swirl angles in the $AR < 2.8$ annular diffusers. Tabs placed normal to the rig axis were less beneficial at disturbing the flow as swirl angle increased from $0^\circ$ to $40^\circ$.

7.1.1 CFD Results Summary

A comparison of simulations using the standard, RNG, and realizable $k$-$\varepsilon$ models produced similar results for swirler angle less than $20^\circ$. The majority of simulations were completed with the R$k$-$\varepsilon$ and SST turbulence models because they have traditionally been selected for diffuser studies. Table 7.1 summarizes the sensitivity of the R$k$-$\varepsilon$ and SST turbulence models to various input parameters.

<table>
<thead>
<tr>
<th>Input parameter</th>
<th>R$k$-$\varepsilon$</th>
<th>SST</th>
</tr>
</thead>
<tbody>
<tr>
<td>Inlet turbulence profile</td>
<td>sensitive</td>
<td>insensitive</td>
</tr>
<tr>
<td>centre-body base curvature (pt. D)</td>
<td>insensitive</td>
<td>sensitive</td>
</tr>
<tr>
<td>geometry changes</td>
<td>better relative comparison</td>
<td>better absolute comparison</td>
</tr>
<tr>
<td>swirl</td>
<td>better outlet axial velocity prediction</td>
<td>better pressure distribution and outlet swirl angle predictions</td>
</tr>
</tbody>
</table>

For all diffusers with $\leq$S20 swirl and no tabs, CFD wall pressure distributions and outlet velocity profiles generally agreed well with experiment. Diffuser inlet total pressure profiles were also in good agreement due to the transition duct inlet boundary condition specification. Discrepancies that
occurred were the result of incorrectly predicting the irreversible total pressure losses in the diffuser, which was a known limitation of the isotropic turbulence models. The error manifested itself in the CFD as a miscalculation of dynamic pressure.

Simulations including centre-bodies with the R$k$-$\varepsilon$ turbulence model showed reasonable agreement in performance to experiment for the $L/\Delta R_t = 12$ axisymmetric annular diffusers with moderate area ratios ($AR < 2.73$) and moderate swirl angles ($S_t < 0.7$). For larger area ratios or stronger swirl, the CFD was less accurate. In particular, SST solutions incorrectly predicted the penalties associated with the larger radial gradients. For example, the back pressure coefficient for $AR = 2.73$ diffusers with $S_0$ swirl was $\sim20\%$ over-predicted with the R$k$-$\varepsilon$ model but $\sim20\%$ under-predicted with the SST model.

For the smaller-angled diffusers ($AR < 2.73$) with tabs, some oscillation was observed in the tab wake. This was a limitation of the steady-state solver used in the present study. That said, the R$k$-$\varepsilon$ turbulence model qualitatively predicted the tab vortex structures correctly as compared to literature [104]. Quantitatively, the tab effect on overall pressure recovery was under-predicted: the CFD predicted a smaller back pressure penalty in comparison to experiment.

### 7.2 Design Advice

Future efforts into the continuing improvement of short annular diffusers will benefit from the contributions of this work. Given that the area ratio yielding maximum pressure recovery is likely $AR = 2.8–4.1$ for the length $L/\Delta R_t = 12$, numerical optimization studies should be completed with say $AR = 4$ constrained. It is anticipated that an $AR = 4$ diffuser is wide enough to benefit from diffuser augmentation that includes pressure recovery. Fluent’s built in mesh optimizer looks promising for shape optimization problems with minimal design changes in the geometry provided that an objective function is known.

Further exploration of VGs, preferentially located at the CB base, is necessary in short annular
diffusers since they do reduce outlet distortion; however, shapes and types other than square tabs should be studied as sizeable separated regions in the tab wakes were observed. Two suggestions were proposed to improve performance:

- Manufacture porous tabs to allow some flow to immediately transport into the wake region.
- Orient the tabs with the flow. Passive alignment may be possible by securing tabs at a pivot point and providing a mechanism to prevent them from rotating all the way around. Steady flow will select a stable stationary position.

Centre-bodies with smooth curvatures should be shaped to provide an initial flow expansion angle of 12–16°. For axisymmetric devices, optimization studies should be completed on coarse 2D grids for economy. Consideration should be given to both design (i.e. 10° inlet swirl) and off-design conditions (CFD not as good at predicting though).

A vortex generator model for tabs can be developed. The stream function \( \psi = V \sin \theta \left( r - \frac{b^2}{r} \right) \) can be used to model the geometrical effects that characterize the tab wake. Analysis of the different configurations determined that dependent variables for a 2D potential flow include inlet velocity, tab width, eccentricity, \( e \), and centre point, \((x_0, z_0)\), of the separated region, and inlet swirl angle. Only \( e \) and \((x_0, z_0)\) require user defined inputs; however, \( e \approx 0.83 \) satisfied most calculated recirculation regions with 4 tabs and linear equations dependent on area ratio were calculated for the major axis length.

### 7.3 Benefits for Industrial Sponsor

Although intellectual property prevented improving an actual device, this academic project will assist future design efforts that are motivated by developing better aerospace exhaust system annular diffusers. The following recommendations are made for the benefit of the Industrial Sponsor:
• A comprehensive review of relevant design methodologies and factors affecting diffuser performance was completed: experiments that evaluated the effects of geometric parameters and inlet swirl obtained the same trends that were described in literature.

• Pursuing vortex generators was encouraged; however, the passive devices must not be arbitrarily attached. Success was not achieved in this project but insight was provided to obtain preferential configurations. Square tabs were less beneficial in swirling flow.

• CFD using steady-state RANS equations was useful for this class of diffuser geometries as a design tool to minimize the variable space. Numerical optimization served as a useful method to identify configurations with improved performance. Coarse grids with wall functions were just as effective at predicting equivalent flow trends with respect to experimentally observed wall pressure distributions and outlet velocity profiles in the $L/\Delta R = 12, 1.91 < AR < 2.73$ annular diffusers. Regardless, experimental work was necessary to instill confidence in the design performance.
References


REFERENCES


REFERENCES


[77] Feldcamp, G. K., Experimental and Computational Evaluation of Strut Losses in a Diverging Annular Diffuser, Master’s thesis, Queen’s University, September 2006. 24
REFERENCES


[82] Ng, B. C. N., Experimental and CFD Study of Effusion Cooling in an S-Bend Diffusing Passage, Ph.D. thesis, Queen’s University, 2013. [24]


REFERENCES


REFERENCES


REFERENCES


Appendix A

Experimental Methods Supplementary

This appendix provides supplementary material for Ch. 3. The bias assessment (App. A.5) was condensed into a paper and presented [162].

A.1 Swirler Profiles

Figures A.1–A.4 show axial and tangential velocity profiles taken 2 cm downstream of the swirler blades and 1 cm downstream of the annular diffuser inlet (with downstream ducting removed). The full 7-hole probe traverse was azimuthally averaged with error bars depicting one standard deviation at the given radius. Left (west) and right (east) side profiles were extracted line traverses from the data set. Higher discrepancies occurred in the tangential velocity profiles with S0 swirl since $v_\theta \approx 0$.

In studies involving swirling flows, it has been proposed to non-dimensionalize pressure by the axial component of dynamic pressure [21, 34, 36, 38]:

$$\langle q_{ax} \rangle = \frac{1}{2} \rho \langle u \rangle^2 \quad (A.1)$$

Equation (A.1) was not implemented since it had no influence on the relationship between configurations other than suggesting that the pressure recovery performance developed by the 40° swirler increased. Calculated ratios of $\langle q_{axz} \rangle/\langle q_t \rangle$ were 0.99, 0.95, 0.76, and 0.98 for the tested swirlers S10, S20, S40, and SCV respectively.
A.2 Three Hole Probe Reduction

The three-hole probes were designed and calibrated to provide axial and tangential velocity components in addition to static and total pressures. For centre port 2 (with side ports 1 and 3), conversion of the raw pressure
A.2. THREE HOLE PROBE REDUCTION

Figure A.3: Section ① ($x = -0.23D_o$) axial velocity profiles.

Figure A.4: Section ① ($x = -0.23D_o$) tangential velocity profiles.

data were achieved by first calculating the coefficients during a calibration experiment. For low angles:

\[
C_B = \frac{p_3 - p_1}{\overline{p} - \overline{p}} \\
C_S = \frac{\overline{p} - p}{p_2 - \overline{p}} \\
C_T = \frac{p_2 - p_0}{p_2 - \overline{p}} \\
\overline{p} = \frac{p_1 + p_3}{2}
\]
and for high angles with port 1 maximum:

\[
C_\beta = \frac{p_3 - p_2}{p_1 - \overline{p}} \quad (A.6)
\]

\[
C_S = \frac{\overline{p} - p}{p_1 - \overline{p}} \quad (A.7)
\]

\[
C_T = \frac{p_1 - p_0}{p_1 - \overline{p}} \quad (A.8)
\]

\[
\overline{p} = \frac{p_2 + p_3}{2} \quad (A.9)
\]

For port 3 maximum, interchange the 1 and 3 subscripts in Eqs. (A.6)–(A.9). During the calibration experiment, the static pressure \( p \) was measured from a wall static pressure tap near the duct outlet and total pressure \( p_0 \) was defined by the port 2 reading when yaw \( \beta = 0^\circ \). The calibration curves shown in Fig. A.5 are typical for all probes at each tested mass-flow.

![3-hole probe calibration curves](image)

**Figure A.5:** 3-hole probe calibration curves. Symbols = raw data, lines = curve fit.

Probe calibration defined calibration constants from forth-order polynomial curve-fits to determine yaw
A.2. THREE HOLE PROBE REDUCTION

angle, static pressure, and total pressure within the three ranges of $\beta$ when the given port read maximum pressure:

\[
\begin{bmatrix}
A_{1,1} & A_{1,2} & A_{1,3} & A_{1,4} & A_{1,5} \\
A_{2,1} & A_{2,2} & A_{2,3} & A_{2,4} & A_{2,5} \\
A_{3,1} & A_{3,2} & A_{3,3} & A_{3,4} & A_{3,5}
\end{bmatrix}
\begin{bmatrix}
1 \\
C_\beta \\
C_\beta^2 \\
C_\beta^3 \\
C_\beta^4
\end{bmatrix}
\begin{bmatrix}
\beta \\
p^* \\
p_0^*
\end{bmatrix}
= \begin{bmatrix}
A_1,1 \\
A_1,2 \\
A_1,3 \\
A_1,4 \\
A_1,5
\end{bmatrix}
\begin{bmatrix}
\beta \\
p^* \\
p_0^*
\end{bmatrix}
\tag{A.10}
\]

For low angles:

\[
p = \overline{p} - p^* (p_2 - \overline{p}) \tag{A.11}
\]
\[
p_0 = p_2 - p_0^* (p_2 - \overline{p}) \tag{A.12}
\]

whereas for high angles, with port 1 maximum:

\[
p = \overline{p} - p^* (p_1 - \overline{p}) \tag{A.13}
\]
\[
p_0 = p_1 - p_0^* (p_1 - \overline{p}) \tag{A.14}
\]

From the calibration test with $p$ and $p_0$ known, $C_S \equiv p^*$ and $C_T \equiv p_0^*$; by inverting Eq. (A.10), $A_{i,j}$ was defined.

Fluid properties for each 3-hole probe were obtained from Eq. (A.10) given the known coefficients $A_{i,j}$ and calculated $C_\beta$. Dynamic pressure, $q$, Mach number, $M$, and the two velocity components, $(u, v_\beta)$, were calculated from:

\[
q = p_0 - p \tag{A.15}
\]
\[
M = \sqrt{\frac{2}{k-1} \left( \left( \frac{p_0}{p} \right)^{k-1/2} - 1 \right)} \rightarrow V = M \sqrt{kR_u T} \tag{A.16}
\]
\[
u_\beta = V \cos \beta \tag{A.17}
\]
\[
v_\beta = V \sin \beta \tag{A.18}
\]

Absolute pressures were required to calculate $M$ with $k$ a constant specific heat ratio and $R_u$ a gas constant.
The probe orientation in the test section meant that \( v_\beta \equiv v_\theta \).

For low yaw angles, Fig. A.5 shows that the central port read highest between approximately \(-25^\circ < \beta < 25^\circ\). At high yaw angles, the opposite port was prone to flow separation. The methodology typically implemented at high angles was to null the 3-hole probe (rotate it to a known angle that accommodates low angle readings during experiment). Instead, an additional step similar in nature to the compressible correction proposed by Gerner [163] was utilized: the calibration procedure was completed multiple times over a range of tip Reynolds numbers based on the probe tip hydraulic diameter \( D_{h,tip} = 2.12 \text{ mm} \) up to \( Re_{tip} \approx 8,700 \). The calibration file with Reynolds number nearest to the experimental conditions total pressure at the centre of the given cross section was selected to reduce the data. Pressure probes become insensitive to incompressible flow at \( Re_{tip} \approx 5000 \) [143].

### A.3 Pressure Transducer Sampling Analysis

Figure A.6 shows moving average pressure and temperature response of 7-hole probe sampling at the centre of the outlet. Sampling was completed at 900 Hz for the first 5 minutes after the blower was started. Time-averaged pressure values were \( p_e = -72.6 \text{ Pa(g)} \) and \( p_{0,e} = 980.0 \text{ Pa(g)} \) with a final temperature of \( T_e = 26.8^\circ \text{C} \). Noise, evident throughout in the pressure signals, was a consequence of sampling data at a location with highly fluctuating flow; however, it took several minutes for the wind tunnel to blow constant temperature air. Typically, the wind tunnel was allowed to warm up for 10 minutes prior to initiating a test.

Figure A.7(a) plots atmospheric pressure offset averages for sampling periods in 1 s increments from several pressure transducers. One data set that took 10 repetitions of 10 s of data at 900 Hz was subdivided into the smaller samples. An argument can be made that greater than a 4 s sampling period was necessary; however, the range in averages was small enough to support that the average from 1 s of data was close enough to the average from a longer sample and damp out any noise that may have been biasing the signal. Figure A.7(b) shows pressure transducer response to the number of samples from a dataset that collected 50 repetitions of 1 s duration at 900 Hz. From a statistical perspective, the student-\( t \) multiplier was 2.228 for ten sets to achieve a two-sided 95% confidence interval (CI). Thirty sets was a better choice since the multiplier reduces to 2.042; however, a compromise with respect to the time required to obtain the sample so ten sets was chosen to provide a reasonable estimate of the average for auxiliary experiment and 3-hole probe measurement repetition.
A.3. PRESSURE TRANSDUCER SAMPLING ANALYSIS

Figure A.6: Blower startup pressure transducer and thermocouple moving averages response.

Figure A.7: Influence of pressure transducer response on time-averaged pressures at atmospheric conditions. Data sets from different pressure transducers.
Figure A.8 shows atmospheric offset averages due to pressure transducer response at different sampling frequencies where each sample obtained one second duration sets times 10 sets of data. The dashed line plots the total amount of time required to collect the measurements. There appeared to be some dependency on sampling frequency. Figure A.9 shows the amplitude spectrum of 300 s of data at 900 Hz taken by the centre port of the 7-hole probe at the outlet centre of a highly turbulent flow with S20 swirl during operation. Spikes occurred at 69 Hz, 124 Hz, 129 Hz, and 255 Hz. Of interest was that a ringing noise was often heard during operation where coincidentally 255 Hz is middle C. The plot confirms that the environment had minimal influence on the sampling signals. Since higher frequency noise was not expected to affect the signal, a sampling frequency of 900 Hz was chosen as it was acquired in a shorter duration than that from lower frequencies.

![Figure A.8](image1.png)

**Figure A.8:** Pressure transducer response to atmospheric conditions at different sampling frequencies. Solid lines = pressure data from different pressure transducers.

![Figure A.9](image2.png)

**Figure A.9:** Single-sided amplitude spectrum of pressure transducer input $y(t)$.  

Figures A.10(a) and (b) were created from the same dataset as Fig. A.9 and show the raw pressure readings over the 300 s and for the first one second collected from an initially stationary probe. Time-averaged pressures for the two figures were -346 and -343 Pa(g) respectively. The difference in averages was negligible such that a one-second sample confidently resulted in a comparable average that was independent of the flow’s transience. For Config. bd ($AR = 6.18$ diffuser), five-second samples were occasionally taken.
A.4 Measurement Uncertainty

Measurements were prone to both bias $B$ and precision $S$ error uncertainty. Uncertainties $U$ were estimated using the root-sum-square model \cite{149, 150}

\[ U = \left[ B_j^2 + (t_{95}S)^2 \right]^{1/2} \]  
(at 95%)

(A.19)

where $t_{95}$ is the student’s $t$ multiplier evaluated based on $N - 1$ degrees of freedom to provide a 95% CI. Table A.1 lists the uncertainty sources considered in the measurements. Biases considered included pressure transducer linearity and hysteresis, calibration curve-fit, and geometrical. Cursory checks in addition to the
data analyzed in App. A.3 confirmed that interference was not present since frequency spikes were not present in the data signal and that the sampling frequency and length were sufficient.

**Table A.1**: Measurement uncertainty sources. Legend: 3=3-hole probe, 7=7-hole probe, w=wall taps

<table>
<thead>
<tr>
<th>Uncertainty</th>
<th>Value</th>
<th>Description</th>
<th>Where</th>
</tr>
</thead>
<tbody>
<tr>
<td>$B_1$</td>
<td>0.2% FS</td>
<td>• linearity and hysteresis</td>
<td>3,7,w</td>
</tr>
<tr>
<td>$B_3$</td>
<td>$p_0 = 0.68%$</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>$\alpha = 0.38^\circ$</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>$\beta = 0.45^\circ$</td>
<td>• probe curve fit calibration</td>
<td>7</td>
</tr>
<tr>
<td>$B_4$</td>
<td>$\alpha = 0.6^\circ$</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>$\beta = 1.3^\circ$</td>
<td>• outlet traverse angle alignment bias, typical</td>
<td>7</td>
</tr>
<tr>
<td>$B_5$</td>
<td>0.08 mm</td>
<td>• outlet radial tolerance</td>
<td>7</td>
</tr>
<tr>
<td>$B_{5a}$</td>
<td>0.5 mm</td>
<td>• traverse coordinate</td>
<td>3</td>
</tr>
<tr>
<td>$B_{5b}$</td>
<td>0.3 mm</td>
<td>• concentricity on $R_i$</td>
<td>3</td>
</tr>
<tr>
<td>$B_{6a}$</td>
<td>0.4%</td>
<td>• tap insertion misalignment</td>
<td>w</td>
</tr>
<tr>
<td>$B_{6b}$</td>
<td>-2%</td>
<td>• tap smoothness</td>
<td>w</td>
</tr>
<tr>
<td>$S_1$</td>
<td>Eq. (A.21)</td>
<td>• measurement repetition</td>
<td>3,7,w</td>
</tr>
</tbody>
</table>

For the 1 psi pressure transducers, the manufacturer recommended a typical uncertainty of $B_1 = 0.1\%$ full scale (FS) (0.5% maximum) be applied to all measurements to account for linearity and hysteresis. Pressure transducers were connected to a water manometer for calibration. Several positive and negative pressures were applied and water manometer height differentials were recorded with a ruler that had a spacing of $\frac{1}{16}$-in, which corresponded to an uncertainty of $B_8 = 2.5$ mm. A linear curve was fitted to the calibration pressure data and the error obtained from the standard deviation of the curve fit and height accuracy was $B_1 = 0.17–0.22\%$ FS. A bias of $B_1 = 27.6$ Pa (0.2% FS) was added to all pressure readings. Furthermore, a repeatability uncertainty of 0.3% FS was recommended; instead multiple sets were taken to serve as a more appropriate measure of repeatability precision, $S_1$.

The probe calibration bias, $B_3$, was defined as the difference between the raw calibration data and the results from using its calibration curve on the raw calibration data. Although the 3-hole probes were also calibrated with a polynomial curve-fit, as shown in Fig. A.5, the uncertainty $B_3$ was several orders of magnitude lower and thus considered negligible.

Unfortunately, the small scale of the model prevented the usage of linear traverse equipment for the 3-hole probes, so they were traversed manually. The bias associated with the probe’s radial location was...
estimated to be $B_{a5} = 0.5$ mm. Additionally, the measurement taken closest to the inner cylinder was prone to the deviation in annulus symmetry and included $B_{a5b} = 0.3$ mm to give a total radial bias of $B_{a5j} = 0.58$ mm. Radial uncertainty propagated into the averaged values using:

$$U_A = 2\pi \sum_{i=1}^{N} B_5 (r_{i+1} - r_i)$$  \hspace{1cm} (A.20)

The uncertainty in the axial locations for all the probes and pressure taps was negligible since the measurements were taken with a height gauge that had an accuracy of plus or minus one-thousandth of an inch.

Since stainless steel wall pressure taps were inserted into thin aluminum ducts, the taps could not be ground down to provide a smooth finish. Although care was taken when inserting the wall taps, it was expected that both the tap alignment on the curved surfaces and possible protrusion led to some error in the measurement. Conservative estimates were obtained from Rayle [164]: $B_{a6} = 0.4\%$ to accommodate an insertion misalignment of $\pm 45^\circ$ from normal and $B_{b6} = -2\%$ on the suspicion that the protrusion could be represented by a burr that could not be felt with the finger. A total geometrical bias for all tap measurements of $B_6 = 2.0\%$ was used.

With the exception of the outlet traverse, all measurements were obtained from a minimum of $M = 10$ sets. Auxiliary experiments were conducted to obtain a repetition uncertainty at the outlet through selecting four locations on the axes of constant radius $r \approx 0.7r_e$ along with the origin. The precision error was quantified by the standard deviation over all of the sets:

$$S_1 = \frac{1}{\sqrt{M/N-1}} \left[ \sum \sum (X_i - \bar{X})^2 \right]^{1/2}$$  \hspace{1cm} (A.21)

\subsection*{A.4.1 Uncertainty Propagation}

Uncertainty of derived quantities for an individual value, $X$, was achieved using:

$$U_X = \sqrt{\sum \left( \frac{\partial X}{\partial v_j} U_{v_j} \right)^2}$$  \hspace{1cm} (A.22)

that was differentiated with respect to each of the dependent variables $v$. However, the uncertainty of an averaged value $\bar{X}$ was more precise by:

$$U_{\bar{X}} = \frac{U_X}{\sqrt{N}}$$  \hspace{1cm} (A.23)
For area-averaged quantities, the uncertainty associated with: Eq. (2.9) was:

\[ U_X = \sqrt{(U_X)^2 + 2 \left( \frac{X U_A}{A} \right)^2} \]  

(A.24)

Whereas, for mass-flow weighted average quantities, velocity propagated into the uncertainty of Eq. (2.10) and gave:

\[ U_{\langle X \rangle} = \sqrt{\left( \langle X \rangle \frac{U_u}{\pi} \right)^2 + 2 \left( \langle X \rangle \frac{U_A}{A} \right)^2 + (U_X)^2 + \left( \frac{X U_u}{\pi} \right)^2} \]  

(A.25)

Note that for the flow measurements, there was a minimal uncertainty of \( B_9 = 2.2^\circ C \) in a thermocouple measuring a near-ambient temperature so the uncertainty in density was neglected since it was at least two orders of magnitude lower than the other uncertainties considered. The velocity uncertainty propagated the incompressible flow relationship:

\[ U_u \approx \frac{\cos \theta \rho}{\rho} \sqrt{\frac{\rho}{2\langle q \rangle}} U_{\langle q \rangle} \]  

(A.26)

where the dynamic pressure uncertainty was:

\[ U_{\langle q \rangle} = \sqrt{U_{\langle (p_0) \rangle}^2 + U_{\pi}^2} \]  

(A.27)

The auxiliary experiments conducted at the outlet provided some insight into the 7-hole probe alignment uncertainty. For the axisymmetric flow, it was assumed that the tangential vectors were zero and the opposing radial vectors were equal and opposite. In Cartesian coordinates without swirl, along the y-axis this meant that \( w = 0 \) and \( -v(-r) = v(r) \). Similarly along the z-axis, \( v = 0 \) and \( -w(-r) = w(r) \). An average of yaw \( \beta = \tan^{-1} \frac{v}{u} \) and pitch \( \alpha = \tan^{-1} \frac{w}{u} \) was then interpreted to be the bias \( B_4 \) that accounted for the misalignment between the duct axis and the 7-hole probe axis. Propagation of repetition uncertainty to pitch and yaw in the auxiliary experiments used:

\[ S_{1,\alpha} = \sqrt{\left( \frac{1}{1 + \left( \frac{u}{v} \right)^2} \right)^2 \left( (wS_{1,w})^2 + \left( \frac{-1}{u^2} S_{1,u} \right)^2 \right) + \left( \frac{-1}{u^2} S_{1,u} \right)^2} \]  

(A.28)

\[ S_{1,\beta} = \sqrt{\left( \frac{1}{1 + \left( \frac{v}{u} \right)^2} \right)^2 \left( (vS_{1,v})^2 + \left( \frac{-1}{u^2} S_{1,u} \right)^2 \right) + \left( \frac{-1}{u^2} S_{1,u} \right)^2} \]  

(A.29)

The average was taken to represent the uncertainty for each individual measurement in the outlet traverse.
A.5 Bias Assessment

The physical nature of several biases were within the user’s control of reducing. Higher tolerances could have been demanded to reduce uncertainty:

- $B_4$ was the discrepancy between the rig axis and 7-hole probe axis.
- $B_5$ addressed the circularity of the outlet shape since the diffuser was welded.
- $B_{5b}$ was a consequence of the component attachment procedure and propagated downstream as a bias in alignment concentricity.
• $B_6$ could have been avoided if the diffuser and pressure tap material were the same, such that the taps could have been ground down and smoothed on the surface after insertion. This did not factor into performance quantification and so did not require dedicated analysis; the bias was assessed through noting any deviations from the multiple taps placed at the same axial locations.

Reducing or eliminating these biases was possible but not practical; instead, the apparatus with its known imperfections were assessed to determine:

• the reliability of the 3-hole probe data
• the influence of intentional misalignments to the collected outlet traverse data

Any biases incurred in tests that quantified performance were unintentional. Data verification was evaluated through comparison of mass flow rates. Appendix E.2.4 further scrutinizes selected diffuser configurations and includes mass flow rate comparisons.

A.5.1 Inlet Traverse Analysis

The three-hole probes traversed the height in increments determined by eight equal areas. A ninth point was obtained by butting the probe against the inner cylinder. Figure A.12 shows the total pressure profiles at the annular diffuser inlet. Since the ninth point did not appear to be biased by edge effects, mass flow rates were calculated using numerical trapezoidal integration. The profile shape was a consequence of maintaining the same outer wall diameter but increasing the inner wall diameter through the transition duct (see Fig. 3.4) and had an area blockage of $A_{R_t} = 4.9\%$.

Both 3-hole traverses at sections $\S$ and $\T$ were completed simultaneously. Results at the diffuser inlet cross section were compared in an early experiment to a traverse of only section $\T$—the manometer data in Fig. A.12—with the 3-hole probe tips at section $\S$ stationed against the outer wall. Larger discrepancies were more likely attributed to the unlikelihood of taking the measurement at the same radial location.

The height of one three-hole probe tip was 6.7% of $\Delta R_t$. This blockage may have biased the measured static pressure. A correction was not implemented due to consistency in the calculated mass flow rates at the four measurement cross sections. For example, mass flow rates evaluated from configurations using OWs a–c were within 5.3% with S0 swirl.
Mass flow similarity was assumed in unswirled flow to compensate for the non-uniformity in the two annular diffuser inlet probe profiles shown in Fig. A.12. A weighting scheme was implemented to obtain an averaged value:

$$X = \chi (w_l X_l + w_r X_r)$$  \hspace{1cm} (A.30)

The left and right side weights were evaluated from the mass-flow in 20° sectors on the left (W) and right (E) quadrants with respect to the total mass flow rate, $\dot{m}_e$, of the outlet data:

$$w_l = \frac{\dot{m}_e}{9 \dot{m}_{e,l}} \quad \quad w_r = \frac{\dot{m}_e}{9 \dot{m}_{e,r}}$$  \hspace{1cm} (A.31)

Although the corresponding outlet total pressure for the configuration considered in Fig. A.12 was higher in the left-hand quadrant and gave $w_l/w_r = 1.4$, the left side station 1 traverse showed lower total pressure measurements due to the left-hand quadrant of the annular diffuser inlet being locally larger in area and so velocity was lower.

To acquire better mass conservation, the derived annular diffuser inlet data, $X$, were scaled by:

$$\chi = \frac{\dot{m}_s}{\dot{m}_{e,o}}$$  \hspace{1cm} (A.32)

---

**Figure A.12:** Total pressure profiles at the annular diffuser inlet (section 1). Manometer readings confirmed the results obtained by the DAQ system.
where \( \dot{m}_{t,o} \) was the annular diffuser inlet mass flow rate prior to scaling. This multiplier was particularly beneficial in cases where the flow bias resulted in both the left and right 20° wedges being higher (or lower) than the average and yielded the averaged line in Fig. [A.12]. Implementing scaling and weighting reduced the average discrepancy between the annular diffuser inlet and outlet mass flow rates for all tests from 6.0% with a standard deviation of 5.5% to 4.1 ± 2.1%. An additional check confirmed that the scaled total pressure did not violate exergy laws within the transition duct.

The choice of weights (for example if \( w_l = 1 \) and \( w_r = 0 \)) had a minimal 2% influence on Eq. (2.1); however, the procedure for implementing Eqs. (A.31) was necessary for establishing higher confidence in the total pressure coefficient obtained from Eq. (2.6). In some cases, the range of \( C_{p0} \) prior to scaling, by choosing only one probe versus the other, was upwards of ±25% of a value assuming equal weighting. Subsequently, the best estimate available with the data collected was to implement Eq. (A.30). Repetition of experiments that were considered to have better alignment reduced the variation of \( C_{p0} \) with respect to a selected configuration with \( Re_l \approx 1.4 \times 10^5 \) and \( M_t = 0.17 \) from an average of 7.0 ± 10.6% without weighting and scaling to 4.6 ± 6.7%.

### A.5.2 Outlet Traverse Analysis

Successive testing of Config. cc (CB|OW, see Fig. 5.1) was done on grids with equal spacing of 6.5, 8.4, and 14 mm that extended one space beyond the wall edge. The extra data point served as confirmation that the traverse did not skip steps since its pressure must be atmospheric in un-swirled flow. Mass-flow weighted average total pressures were within 1.7% and mass-averaged axial velocities were within 0.2% between the three grids. Although the coarser grids yielded minimal difference in all evaluated properties, grids nearest to 7 mm spacing were primarily implemented since there was not a significant time expense to complete a traverse (about 30 min for OWb).

### Axis Alignment Study

This study involved tilting the outer wall when it was fastened onto the annulus—there was a rotational offset between the CB and OW axes, i.e., . Figures [A.13(a)], [A.13(c)] show outlet profiles from experiments using Config. cc (CB|OW) completed in succession. Annular diffuser inlet properties were \( Re_t = 1.5 \times 10^5 \) and \( M_t = 0.17 \) (\( \langle V_t \rangle = 60 \text{ m/s} \)) with an air static temperature of \( T_t = 300 \text{ K} \). Minor adjustments were made
to the position of the transition duct and diffuser outer walls with respect to the annulus by inserting 3.2 mm (1/8-in) thick washers at selected locations between the annulus and transition duct flanges. In comparison to the best-aligned case in Fig. A.13(b) whose outlet centre was within 1.9 mm of the centre-body axis, the outlet centres of Fig. A.13(a) and Fig. A.13(c) were 5.9 mm lower and 6.8 mm (0.05\(D_e\)) higher respectively due to an angular bias between the annulus axis and diffuser outer wall axis.

Figure A.13: Total pressure contours and velocity vectors of outlet profiles where only changes to outer wall axis alignment were made.

The better alignment of Fig. A.13(b) improved the annular diffuser inlet height to \(\Delta R_t = 19.3 \pm 0.1\) mm whereas the shifted outlets had a maximum height difference of \(\sim \pm 0.3\) mm (2%). There were difficulties associated with obtaining a confident reading of \(\Delta R_t\) due to the fact that the inner cylinder was not completely rigid, owing to the longer moment arm stemming from the airfoil supports.

Figures A.13(a)–(c) show the dramatic difference in flow contours—particularly the magnitude of the high region—and vector directions. Asymmetric mass imbalances were \(w/w_r = 0.95, 1.02, \) and 0.93 and area blockages were \(A_{B,e} =40\%, 37\%, \) and 39% respectively. Diffuser inlet mass-flow discrepancies with respect to the outlet were -3.9\%, -5.3\%, and -6.2\%. Millimetre discrepancies may seem insignificant but since \(\Delta R_t\) was small, the bias in concentricity at the annular diffuser inlet was 3%. Fortunately, the effect was mitigated in averaging since the outlet total pressure \(\langle p_{0,e} \rangle\) was virtually identical in Figs. A.13(a) and (c) and within 3.2\% of Fig. A.13(b).

Figures A.14(a)–(b) show the respective CB and OW pressure distributions for the best-aligned and
bottom bias tests. The overlapping data between the two cases indicated that static pressure was insensitive to the alignment disparity. Furthermore, the fact that the left-side (pressure tap on south-west axis) and right-side (north-east) quantities also overlapped, confirmed that the symmetry of the annular diffuser dictated the pressure distribution regardless of whether or not the incoming flow was concentric. The local discrepancy that occurred at the first taps in Fig. A.14(b) after $x = 0$ was attributed to the flange connection not being completely smooth where the sides were subjected to a step differentiation—this feature did not impact the static pressure further downstream.

![Figure A.14:](image)

The variation in $\frac{p}{\langle q_t \rangle}$—that was determined from the 3-hole probes located at $x = -0.22D_o$—between the two cases in Figs. A.14(a)–(b) was less than 0.1%. In comparison, the pressure difference between the annular diffuser inlet left-side and right-side measurements was upwards of 5%. The discrepancy associated with alignment was negligible considering that the pressure tap biases $B_1$ (67%) and $B_6$ (31%) combined to give an absolute uncertainty of $\sim 30$ Pa on measurements reading near-atmospheric conditions.

Table A.2 distinguishes the sources of uncertainty by property used to calculate the objective uncertainties for Config. cc (best alignment). The uncertainties obtained at the annular diffuser inlet were quite acceptable and were dominated by pressure transducer linearity and geometrical errors. The absolute contribution from the error associated with the resulting flow asymmetry was not substantial enough to detriment confidence in the derived flow profile.

Outlet uncertainty values given in Tab. A.2 were significantly higher and dominated by the error associated with the auxiliary experiment repetition. This was attributed to transient effects in the flow. For example,
Table A.2: Uncertainty contribution by property in axis alignment study for best Config. cc (outlet data shown in Fig. A.13(b)). The tabulated relative quantities of the error sources were not propagated.

<table>
<thead>
<tr>
<th>Quantity</th>
<th>Uncertainty, % (95%)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>U</td>
</tr>
<tr>
<td>⟨p₀,t⟩</td>
<td>1.7</td>
</tr>
<tr>
<td>p₀</td>
<td>1.3</td>
</tr>
<tr>
<td>uₜ</td>
<td>1.4</td>
</tr>
<tr>
<td>⟨p₀,e⟩</td>
<td>13.5</td>
</tr>
<tr>
<td>pₑ</td>
<td>5.4</td>
</tr>
<tr>
<td>αₑ</td>
<td>0.7°</td>
</tr>
<tr>
<td>βₑ</td>
<td>1.8°</td>
</tr>
</tbody>
</table>

the repeated total pressure measurements for the reference case taken at the outlet centre had a standard deviation of 61%. The outlet traverse of this configuration was also completed with a two second sampling rate and yielded similar uncertainties.

Axis Shift Study

Figures A.15(a)–(c) show total pressure contours obtained when there was a smooth OW transition versus tests completed with the OW shifted 2 mm left and 2 mm right with respect to the transition duct, i.e., the CB and OW axes were translated, . Config. nb (CB|OW) with the SCV swirler was used. In comparison to the study that produced Figs. A.13(a)–(c) where any shift in axis translation was more-or-less unintentional, the duct was intentionally shifted to obtain Figs. A.15(a) and (c). Performance of the shifted ducts was worse: roughly 4% in C₝, 3% in γ, and 8% in Cₚ₀. Therefore, some amount of precision was necessary during the apparatus assembly.

Influence of Upstream Blockages

Figures A.16(a) and (c) show total pressure contours and axial velocity profiles from a test completed that extended the 3-hole probes (axial locations shown in Fig. 3.4) into the flow that each blocked 0.9% of the area at section due to the probe’s 4.0 mm (5/32 in) shaft diameter. Tests were completed with Config. bb (CB|OW) and S0 swirl. The influence was quite apparent at the outlet and produced total pressure and velocity
deficits along the y-axis in an otherwise uniform flow. No other outlet profiles, for example Fig. A.15(b), showed any indication of deficits when the 3-hole probes were stationed against the outer wall.

The axial vorticity contours in Fig. A.16(b) show that the probes created sizeable counter-rotating vortex pairs at the outlet due to diffusion. Erroneous gradients were calculated along the positive y-axis due to the fact that post-processing was completed on a cylindrical grid. The circled high-vorticity zones correlated well with the total pressure distortions that were caused by the probe necks. Two additional pairs were evident: the smaller coherent structures may be attributed to the probe tips. The remainder of the outlet was considered irrotational.

Similarly to the flow distortions caused by duct misalignment, probe obstructions had minimal impact on the outlet averaged properties. When compared to results with the probes retracted, there was < 0.1% change to $C_b$, a 0.24% improvement in $\gamma$, and a 6.6% loss in $p_{0,e}$; however, the respective uncertainties in $\gamma$ and $p_{0,e}$ were 3.8% and 3.6%.
A.5. BIAS ASSESSMENT

Figure A.16: Influence of 3-hole probes in flow on the outlet profile. Probes were fully extended into the flow at section 1 during the outlet traverse.
Appendix B

Geometry Optimization

An evolutionary algorithm with seven design variables and three competing objectives was a systematic approach implemented to reduce the decision maker’s bias in selecting the annular diffuser geometry. The manufactured configurations were selected from the set of non-dominated solutions. This study was published as a conference proceeding and presented [165].

B.1 Literature Review

Increased interest in numerical shape optimization has been a result of greater computational power and better evolutionary algorithms. It should be emphasized that wind tunnel testing is still important in the design process, but in a complementary role to CFD. Table B.1 lists strengths and weaknesses from the two methods [59]. When used in tandem, CFD can narrow the design space (screening a few geometries from a pool of many possibilities) and experiment can give confidence in the performance of the chosen design—this achieves significant development cycle time reductions. Experience with the CFD methodology, when applied to similar problems, negates the need for experimental validation.

The bulk of previous numerical optimization efforts have focused on airfoils defined by a degree-of-freedom curve with usually one or two competing objectives [166, 169]; however, the application of genetic algorithms to other devices is gaining popularity. Examples of shape optimization on more complex flows include a flame-holding bluff body [170], microchannel heat sink [171], pump impeller [172], rocket-ejector that simultaneously optimized both rocket and air flowpaths [173], and diffuser [174, 175].
Table B.1: Experiments and CFD strengths and weaknesses.

<table>
<thead>
<tr>
<th></th>
<th>Experiments</th>
<th>CFD</th>
</tr>
</thead>
<tbody>
<tr>
<td>strengths</td>
<td>- deals with real fluid</td>
<td>- can test full-scale models</td>
</tr>
<tr>
<td></td>
<td>- produces global data over a far greater range</td>
<td>- beneficial for design improvement</td>
</tr>
<tr>
<td></td>
<td>- better suited for validation and database building since there is more confidence in the calculated performance coefficients</td>
<td>- rapid and cheap to carry out simulations</td>
</tr>
<tr>
<td></td>
<td></td>
<td>- good for inverse design or optimization</td>
</tr>
<tr>
<td>weaknesses</td>
<td>- flow compromised due to apparatus mounting mechanisms and instrumentation</td>
<td>- accuracy limited by numerical model assumptions</td>
</tr>
<tr>
<td></td>
<td>- large capital costs</td>
<td></td>
</tr>
</tbody>
</table>

Insights regarding the limitations of CFD optimization have been made. Hu et al. [176] used a genetic algorithm to optimize an S-shaped inter-turbine duct that modelled the endwall geometry with 11-point Bezier curves and selected a solution with minimum total pressure loss. Simulations were completed with the SST turbulence model and conclusions suggested to limit the use of optimization to geometries with less aggressive parameters such that the numerical solutions correctly capture the flow physics.

Jirásek [11, 76] used CFD to optimize vortex generators in an S-duct diffuser. Since unreplicated design was implemented, different diagnostic techniques along with a number of verification runs were recommended to check the surface model response accuracy. Similar computational studies were completed by Wendt and Dudek [75] and Anderson and Gibb [102]. Flow parameters were not included since VG height depended on boundary layer thickness for example. To avoid mesh generation difficulties associated with embedding the VG shape, a vortex generator model was implemented that sought to match the effect. When dimensional variables were implemented, the study failed. Wind tunnel testing helped establish confidence in the CFD optimization methodology.

B.2 Problem Description

Designing an annular diffuser for turbine discharge is device specific. Sovran and Klomp [1] compiled a list of manufactured diffusers for turbomachinery applications that had inlet radius ratio $R_i/R_o$ ranging from 0.48
to 0.79, length to inlet height $L/\Delta R$ of 1.0–9.8, and area ratios of 1.3–4.1 with an average of $AR = 2.2$. The devices used in their study had an $R_i/R_o$ of either 0.55 or 0.70 with length and area ratio ranges similar to those already manufactured. Additional literature included in Ch. 2 analyzed annular diffusers with $R_i/R_o = 0.58–0.83$, $L/\Delta R = 3.4–10$ and $AR = 1.3–4.3$ [7, 9, 16, 17, 33, 34, 36–38, 51].

Design requirements used to develop the annular diffusers tested in this work are summarized in Tab. B.2. The inlet radius ratio was defined as $R_i/R_o = 0.75$. Given that the outer wall converged, this ratio prevented a decreasing area distribution determined by the relationship:

$$A = \pi (R_o^2 - R_i^2) = \pi R_i^2 \quad \Rightarrow \quad \frac{R_i}{R_o} = 0.71 \quad (B.1)$$

A length of $L = 1.5D_o \equiv 12\Delta R_i$ was chosen as a representative value to classify this device as short and hence meet the space constraints in typical aerospace applications. The seven free variables included five that defined the centre-body shape consisting of three flat panels $DE$, $EF$, and $FG$ and two variables to define the outer wall $ST\cdot TU$ profile. Figure B.1 illustrates that the annular diffuser exhaust system had a conical expansion followed by a solid fully open outer diameter diffusing section with divergence angle $2\alpha_{TU} = 12^\circ$. McDonald and Fox [138] observed that $12^\circ$ was an upper limit for an unstalled conical diffuser.

The concentric annulus was modelled with an inner diameter $D_i \equiv 2R_i = 16.2$ cm and outer diameter of $D_o \equiv 2R_o = 21.6$ cm (8.5 in). (Subsequent simulations of the manufactured components were completed using $D_o = 6$ in to match the wind tunnel dimensions.) Note that for this optimization study, the annular diffuser inlet section $\bullet$ was located at plane $DS$.

### Table B.2: Exhaust system design requirements

<table>
<thead>
<tr>
<th>Function</th>
<th>Maximize performance</th>
</tr>
</thead>
<tbody>
<tr>
<td>Fluid properties</td>
<td>Unswirled air with $Re_t = 8.5 \times 10^4$ and $M_t = 0.23$</td>
</tr>
<tr>
<td>Constraints</td>
<td>$R_i/R_o = 0.75$, $L = 1.5D_o$ and $2\alpha_{TU} = 12^\circ$</td>
</tr>
<tr>
<td>Objectives</td>
<td>1. Maximize diffuser pressure recovery, Eq. (2.1)</td>
</tr>
<tr>
<td></td>
<td>2. Maximize outlet velocity uniformity, Eq. (2.4)</td>
</tr>
<tr>
<td></td>
<td>3. Minimize total pressure loss → maximize Eq. (2.6)</td>
</tr>
<tr>
<td>Free variables</td>
<td>Seven geometric variables defined in Fig. B.1</td>
</tr>
</tbody>
</table>
B.2. PROBLEM DESCRIPTION

Figure B.1: Annular diffuser schematic. section ① denotes the annular diffuser inlet at plane DS and ⑤ the diffuser outlet with radius \( r_U \).

B.2.1 Constraints

A preconceived notion for preferential designs led to the minimum and maximum limits listed in Tab. B.3. The variables selected and relative nature in which they were defined was done to prevent CB shapes that avoided decreasing flow area or concave curvature through requiring \( x_E < x_F < x_G \). Geometric relations were implemented to define the CB shapes from the flow angles \( \alpha_1 = \alpha_{ST} - \alpha_{DE} \) and \( \alpha_2 = \alpha_{ST} - \alpha_{EF} \), radial values \( r_E, r_F \), and axial value \( x_G \). The CB variable limits accommodated a range of geometries from plug nose to conical. The maximum CB length was assigned in acknowledgement of structural limitations.

<table>
<thead>
<tr>
<th>Variable</th>
<th>Minimum</th>
<th>Maximum</th>
</tr>
</thead>
<tbody>
<tr>
<td>( \alpha_1 )</td>
<td>( 5^\circ )</td>
<td>( 20^\circ )</td>
</tr>
<tr>
<td>( \alpha_2 )</td>
<td>( \alpha_1 + 5^\circ )</td>
<td>( \alpha_{ST} - \alpha_{EG} )</td>
</tr>
<tr>
<td>( r_E )</td>
<td>( 0.55 r_D )</td>
<td>( 0.95 r_D )</td>
</tr>
<tr>
<td>( r_F )</td>
<td>( 0.1 r_D )</td>
<td>( 0.9 r_F )</td>
</tr>
<tr>
<td>( x_G )</td>
<td>( 0.2 D_o )</td>
<td>( 0.8 D_o )</td>
</tr>
<tr>
<td>( x_T )</td>
<td>Eq. (B.2)</td>
<td>( D_o )</td>
</tr>
<tr>
<td>( r_T )</td>
<td>( R_i )</td>
<td>( R_o )</td>
</tr>
</tbody>
</table>
Given the requirement to have solid fully open outer diameter axisymmetric flow exhausting the diffuser, the initial outer wall panel $ST$ could initially be converging (i.e. $r_T < R_o$). To guarantee that solutions existed where cross section area did not decrease above the CB:

$$(\sin 45^\circ)^3 (x_{T,\text{min}})^2 + 2 R_i x_{T,\text{min}} \sin 45^\circ - (R_o^2 - R_i^2) = 0 \quad (B.2)$$

### B.3 Methodology

Evolutionary algorithms offer a systematic approach that reduces the decision maker’s bias in identifying the preferential nondominated solutions. The algorithm principle chooses parents from existing solutions to create children (new solutions) with better characteristics. An iterative process occurs that eventually converges towards an optimized solution (or solutions). Different methodologies have been developed to carry out this procedure where some are considered more efficient than others; however, the objective for the implementation of a genetic algorithm to the present problem was for proof of concept that preferential solutions existed and that the evolutionary process developed solutions with better objectives; therefore, only one algorithm was considered. Obtaining solutions with CFD was a cost-effective means for simulating many geometries and choosing configurations that suited the project requirements.

The non-dominated sorting genetic algorithm II proposed by Deb, Pratap, Agarwal, and Meyarivan [177, 178] was implemented to compare solutions during the iterative process. In the case of maximizing three competing objectives, the non-dominated solution set (also known as a Pareto front or trade-off surface) has the characteristic whereby no other solutions exist at a given solution in the upper right-hand quadrant containing the theoretical ideal solution with maximum values for all objectives [179].

The algorithm first randomly generated $N = 8$ geometries (parents). Steady-state solutions were obtained using the CFD software ANSYS Fluent 13.0 [84]. Fine grids were constructed on 30° wedges and flow properties were evaluated using the RANS equations with the realizable $k-\varepsilon$ turbulence model and enhanced wall treatment. Additional information describing the CFD procedures is in Ch. 4.

A per-generation automated process as depicted in Fig. B.2 was performed to decrease the time required to complete the repetitive tasks. The genetic algorithm was supplied with the objective quantities (Eqs. (2.1), (2.4), and (2.6)) calculated from converged CFD solutions for the $N$ children from Generation (i). These $N$ children were sorted with respect to the $N$ parents from Generation (i-1) by non-domination level (Pareto
front rank) and crowding distance (nearness of adjacent solutions on the same Pareto front).

The front rank check algorithm accommodated minimizing or maximizing objectives. It was more convenient to order solutions to seek out maximum objective values but solutions to the left of a local maximum were included on the suspicion that they could produce new children with higher maximums for the secondary objectives. The range was truncated as necessary. First, the $2N$ solutions were ordered from minimum to maximum using the first objective (Eq. (2.1)). Differences between adjacent solutions $X_j - X_{j-1}$ and maximums were calculated for the subsequent objectives. The algorithm kept positive differences up to the objective(k) maximum and negative differences beyond. If a solution satisfied the differencing requirements for all objectives(k), then it was declared to lie on the given rank (first). If solutions were rejected, the subsequent rank (two and so on) was determined from this rejected set in a similar fashion. The process continued until all solutions were assigned a rank.

The set of solutions on a given rank were again sorted from minimum to maximum according to the first objective. The crowding function for the $j^{th}$ solution:

$$crowd(j) = \frac{\sum_{k=1}^{N_{obj}} |X_{j+1,k} - X_{j-1,k}|}{X_{\text{max},k} - X_{\text{min},k}}$$

assigned intermediate distances to all solutions for objective quantity, $X$, and number of objectives, $N_{obj} = 3$. Endpoints were given a value $crowd(j) = N_{obj}$ and local maximums for secondary objectives were given $crowd(j) = 0.5N_{obj}$. All ranks proceeded in this fashion. It was with this crowding function that the range could be truncated: for example, solutions with objective 1 below 0.5 were given a value of $crowd(j) = 0.01$ and the minimum was selected as the first solution with objective value greater than 0.5.
The set of $2N$ solutions were ordered first by Pareto rank and then crowding distance. The top $N$ solutions became the $N$ parents for Generation (i). Over-riding the endpoints and local maximums guaranteed that those solutions remained in the set. The $N$ children for Generation (i+1) were created from a stochastic selection process. First, the parents were weighted using a roulette wheel function:

$$\omega = \frac{\rho}{\sum \rho} \quad \rho = rank^{-2} + \frac{crowd/n_{obj}}{n_{obj}} \quad (B.4)$$

that gave preference to less crowded solutions on the first front. Two parents, $a$ and $b$, were chosen randomly using a tournament selection approach based on the weighted values. Depending on the range of the probability parameter—a random value to declare how children were created—either binary tournament selection, recombination, or mutation occurred. If the probability parameter was less than the probability of survival $p_{surv}$, a random variable below 0.5 determined that the design variable took on the value of parent $a$ and random variable above 0.5 gave the value of parent $b$. The child’s design variable was averaged using:

$$v = v_a + rand (v_b - v_a) \quad (B.5)$$

if the probability parameter fell within $p_{surv}$ and $p_{surv} + p_{avg}$. Mutation occurred for probability parameter greater than $p_{surv} + p_{avg}$ and meant that the design variable could be any number between zero and unity. Fractions were implemented to define the variables since equations:

$$x = x_{min} + v (x_{max} - x_{min}) \quad (B.6)$$

were defined to satisfy the ranges given in Tab. B.3. All seven design variables for each child were specified in this fashion.

This genetic algorithm possessed both exploration and exploitation selection characteristics:

- Exploration: the genetic algorithm was capable of searching new regions of the search space. This was achieved if a variable was recombined or mutated. Equation (B.5) may be overly explorative to the extent of being considered a random search function since the resolution was $10^{-8}$ for a possible $10^{56}$ solutions given the 7 variables. But since the objectives were competing, it was assumed that upon choosing optimized parents that lie on the Pareto front, the solution set was better filled in by children whose variables were between the parent values.
B.4. COMPUTATIONAL VERIFICATION

• Exploitation: the ability to focus on a given local area of the search space by means of variable survival. The risk existed that a population might not be diverse enough and the algorithm would become trapped on a set of sub-optimal solutions.

The benefits of user interaction between generations allowed the possibility to make modifications to the genetic algorithm. For the first twenty generations, probability of survival was $p_{\text{surv}} = 0.6$ and averaging probability was $p_{\text{avg}} = 0.35$. Furthermore, in the event of averaging, each of the seven variables used to determine a child could have different parents. These values were specified on the assumption that the variable ranges were satisfactorily defined.

At the 20th generation, it was observed that the best geometries excluded solutions with $r_T = R_o$ and $\alpha_1 > 13^\circ$, for example, so manual overrides were inserted. After the 20th generation, $p_{\text{surv}} = 0.1$, $p_{\text{avg}} = 0.85$, and only two parents were selected to determine all the variables for one child (unless mutation occurred). These changes meant solutions were more likely to enter the non-dominated set and then get discarded in a subsequent generation by a better solution. The benefits of a higher averaging probability was particularly useful for this study given the relative nature in which the variable ranges were defined. The process terminated after thirty-five generations.

B.4 Computational Verification

Three-dimensional grids were constructed for this optimization study for two reasons: (i) A previous awareness of a flaw in 2D solvers (that has since been rectified in Fluent); and (ii) having no preconceived notion on the magnitude of secondary flows.

Wedge Angle

Configurations aa, bb, and cc (CB|OW) from the non-dominated solution set were converged on grids with a 60° wedge angle (see Fig. B.5 for geometry). In comparison to the 30° wedge medium grid results listed in Tab. B.4 objective quantities were all within 2%. Owing to the higher radial pressure gradients present in Config. aa, the maximum axial vorticity measured on several cross sections in the diffuser was 1200 s$^{-1}$. Given the absence of any appreciable high $x$-vorticity regions, this flow did not have significant secondary flows to verify the use of 2D grids.
Grid Study

Table B.4 presents results for three configurations (see Fig. B.5 for geometry profiles) from the non-dominated solution set after the 35th generation that underwent a grid independence study. The three grids were created with geometrically similar cells: a coarse grid with $dx = 0.0022$ m, a medium grid with $dx = 0.0016$ m, and a fine grid with $dx = 0.0012$ m.

Table B.4: Uncertainty in performance parameters for genetic algorithm grid independence study on an annular diffuser with respect to the medium grid. GCI_c are results of the medium grid compared to the coarse grid and GCI_f are results of the medium grid compared to the fine grid.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Cells, $10^6$</th>
<th>$C_b$</th>
<th>$\gamma$</th>
<th>$C_{f0}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Config. aa</td>
<td>medium grid</td>
<td>1.43</td>
<td>0.61</td>
<td>0.95</td>
</tr>
<tr>
<td>GCI_c, %</td>
<td>0.53</td>
<td>0.50</td>
<td>0.23</td>
<td>2.02</td>
</tr>
<tr>
<td>GCI_f, %</td>
<td>3.06</td>
<td>0.94</td>
<td>0.35</td>
<td>4.86</td>
</tr>
<tr>
<td>Config. bb</td>
<td>medium grid</td>
<td>1.44</td>
<td>0.69</td>
<td>0.92</td>
</tr>
<tr>
<td>GCI_c, %</td>
<td>0.54</td>
<td>0.53</td>
<td>0.42</td>
<td>2.80</td>
</tr>
<tr>
<td>GCI_f, %</td>
<td>3.11</td>
<td>0.83</td>
<td>0.55</td>
<td>5.62</td>
</tr>
<tr>
<td>Config. cc</td>
<td>medium grid</td>
<td>1.42</td>
<td>0.77</td>
<td>0.77</td>
</tr>
<tr>
<td>GCI_c, %</td>
<td>0.53</td>
<td>1.57</td>
<td>1.97</td>
<td>8.59</td>
</tr>
<tr>
<td>GCI_f, %</td>
<td>3.15</td>
<td>1.85</td>
<td>2.61</td>
<td>13.35</td>
</tr>
</tbody>
</table>

Evaluation of grid numerical uncertainty followed the grid convergence methodology outlined in Roache [180] and Freitas et al. [181]. Error estimates were calculated using a grid convergence index:

$$GCI_c = F_s \frac{|X_2 - X_1|}{1 - r^p}$$  \hspace{1cm} (B.7)

$$GCI_f = F_s \frac{r^p(X_2 - X_1)}{1 - r^p}$$  \hspace{1cm} (B.8)

where $r = dx_2/dx_1$ is the refinement factor between the coarser grid (subscript 2) and finer grid (subscript 1) and $X$ is an objective quantity [180]. Table B.4 shows conservative estimates for the numerical uncertainty in performance parameters on the medium grid that used an order of accuracy of $p = 2$ and a factor of safety of $F_s = 1.25$ (for a 95% CI). Based on this analysis, the back pressure coefficient of Config. aa was
Although values reported in Tab. B.4 suggest convergence to similar solutions, the uncertainty was likely on the high side for a typical CFD study. On an absolute scale, the conclusions from the genetic algorithm remained unchanged. Since convergence economy was a necessary requirement given the scope of the project, larger grid uncertainties resulting from the medium grid design was an unfortunate consequence of simulating 3D grids that resolved the boundary layers.

### B.5 Optimization Study Results

Figure B.3 compares total pressure loss coefficient to back pressure coefficient for all solutions generated. Several pop-outs depict geometry profiles for the corresponding objective quantities. A linear trend occurred whereby lower total pressure loss was proportionate to higher static pressure recovery; however, the clustering of solutions in the top-right corner shows that the geometry variations did, in fact, develop a set of non-dominated solutions. If not intuitive, the performance of the pop-out geometries confirms the utility of CFD to reduce the variable search space in a design of experiment.

Figures B.4(a) and (b) present objective quantities for solutions generated within the clipped ranges shown. The results gave strong evidence that maximizing pressure recovery was a competing objective with respect to both maximizing outlet velocity uniformity and minimizing total pressure loss. Competition occurred because of the two factors that influence pressure recovery: (i) an increase in area that can be assumed isentropic for attached flow and (ii) large scale turbulent mixing that was affected by the centre-body shape.

The inconsistent nature of the square locations emphasized by the segmented nature of the dashed line defining the non-dominated front in Figs. B.4(a) and (b) meant that the ideal Pareto solution set had not been achieved. Of the 31 solutions that had been identified as non-dominated solutions after the 35th generation, all were created after the 20th generation with 18 created after the 30th generation—this validated the evolutionary process of the genetic algorithm whereby parents produced children with better characteristics.

Figure B.5 plots the geometry profiles for the non-dominated solutions after the thirty-fifth generation. The most apparent variation was with respect to $r_T$ where $r_T \to R_i$ corresponded to solutions with higher values of $\gamma$ and $C_{p_0}$ whereas $r_T \to R_o$ gave solutions with higher $C_b$. The fact that all of the non-dominated
Figure B.3: Optimization study objective space showing all solutions comparing total pressure loss to pressure recovery. Results show all evaluations during 35 generations of the genetic algorithm with selected geometries shown (pop-out ratio \( \frac{x}{r} = 0.5 \)). The suits define original parents where: ♣ = minimum values provided in Tab. B.3 to produce a plug nose centre-body, ♦ = maximum values for the CB that resulted in a conical shape but minimum \( r_T \) and \( x_T \), and ♠ = maximum values provided in Tab. B.3.

solutions had \( x_T = D_o \), the maximum constraint, suggested that there may be additional performance improvements by further reducing the solid diffuser length \( TU \), through considering \( x_T > D_o \).

The CB profiles in Fig. B.3 indicate preference to a paraboloid shape with length \( x_G = 0.74D_o \) and initial expansion angle of \( \alpha_1 = 13.2–14.8^\circ \). Profiles that gave better \( \gamma \) and \( C_{p_0} \) were more conical with panel \( DE \) length of \( l_{DE} \approx 0.34D_o \) whereas higher \( C_b \) was obtained with a more blunt-shaped end panel \( FG \) and longer \( l_{DE} \rightarrow 0.67D_o \). Interestingly, the radial location of point \( F \) remained constant at \( r_F \approx 0.17D_o \); to confirm that this was not a consequence of the initially high survival probability, approximately 70% solutions generated by the genetic algorithm had a value of \( r_F \) greater than 10% away from 0.17D_o. Trends were not obvious for panel \( EF \) other than all non-dominated solutions had an expansion angle in the range \( 20^\circ \leq \alpha_2 \leq 35^\circ \).
B.5. OPTIMIZATION STUDY RESULTS

![Optimization study objective space](image)

**Figure B.4:** Optimization study objective space. The circles denote discarded solutions whereas the 31 squares define non-dominated solutions after the 35th generation. Solid diamonds are solutions with: Config. aa = maximum $\gamma$ and $C_{p0}$, Config. bb = best solution with equally weighted objectives, and Config. cc = maximum $C_b$. 

(a) outlet velocity uniformity vs. pressure recovery

(b) total pressure loss vs. pressure recovery
Figure B.5: Non-dominated solution geometry profiles after the 35th generation. Similar cyan or magenta symbols for points E, F, T, and U denote intermediate solutions on the non-dominated front.
Appendix C

CFD Executable Scripts

Table C.1 lists text files included in this appendix to generate a 2D coarse grid in *ICEM CFD*, simulate the mesh in *Fluent*, and output a text file with the performance parameters. Scripts are based on running release 15.0 ANSYS software. These scripts are functional but simplified from the implemented files. The generated grid only models the diffuser section.

<table>
<thead>
<tr>
<th>file name</th>
<th>purpose</th>
</tr>
</thead>
<tbody>
<tr>
<td>Wallpts.dat</td>
<td>formatted point data that defines the diffuser geometry</td>
</tr>
<tr>
<td>BCinlet.prof</td>
<td>diffuser inlet boundary condition mass flux profile</td>
</tr>
<tr>
<td>batchme</td>
<td>bash linux script to generate mesh and run Fluent in background</td>
</tr>
<tr>
<td>postme</td>
<td>bash linux script to obtain performance parameters $C_b$, $\gamma$, and $C_{p0}$ in an output file</td>
</tr>
<tr>
<td>coarsegrid2D.tcl</td>
<td>load script file in ICEM CFD to generate mesh</td>
</tr>
<tr>
<td>flrke2DS0.jou</td>
<td>load journal file in Fluent to define project and obtain solution</td>
</tr>
<tr>
<td>fortransorter.f90</td>
<td>post process Fluent output to calculate performance parameters</td>
</tr>
</tbody>
</table>

To execute:

1. create files and place on linux cluster in working folder (rootdir).
2. execute command: bash batchme
3. execute command: bash postme
The solution converged in 660 iterations with performance quantities listed in Tab. C.2.

<table>
<thead>
<tr>
<th></th>
<th>$C_b$</th>
<th>$\gamma$</th>
<th>$C_{p_0}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>included script (no plenum)</td>
<td>0.645</td>
<td>0.959</td>
<td>-0.058</td>
</tr>
<tr>
<td>$Rk-\varepsilon$-c (with plenum)</td>
<td>0.668</td>
<td>0.941</td>
<td>-0.051</td>
</tr>
</tbody>
</table>

### C.1 Geometry Input File

This file provides Cartesian coordinates ($x, y, z$) for the symmetry plane of the annular diffuser geometry:

- first five points define inner wall $D, E, F, G, H$
- last three points define outer wall $S, T, U$

Text for Wallpts.dat used to create Config. bb:

0.00000000 0.05715000 0.00000000
0.05467464 0.03817541 0.00000000
0.08982307 0.02189779 0.00000000
0.11663510 0.00000000 0.00000000
0.22888575 0.00000000 0.00000000
0.00000000 0.07645400 0.00000000
0.15303500 0.06273800 0.00000000
0.22888575 0.07018733 0.00000000

### C.2 Boundary Condition Input File

This file contains experimental results obtained from Config. bb with S0 swirl that operated at $Re_t = 1.4 \times 10^5$ and $M_t = 0.16$. Contents of BCinlet.prof:

```
((inletbc radial 11)
(r
 0.05715001
 0.05795000
 0.05854328
```
C.2. BOUNDARY CONDITION INPUT FILE

```
0.06123481  
0.06381292  
0.06629084  
0.06867941  
0.07098766  
0.07322318  
0.07539245  
0.07645399  
)  
(axial-velocity  
56.36849329  
50.07189128  
56.14062116  
57.98205448  
58.99679525  
59.05634335  
58.24672328  
56.76277783  
55.47588141  
52.43218940  
50.40069476  
)  
(total-pressure  
75.69755289  
502.56895843  
819.14117302  
904.01475432  
940.37117219  
931.25682773  
867.52793804  
762.75018763  
649.43246000  
468.23694167  
357.23790848  
)  
(mass-flux-act  
65.71122460  
58.37099951  
65.44558407  
67.59222364  
68.77515145  
68.84479195  
67.90075967  
66.17085938  
64.67066778  
61.12250253  
58.75430015  
)  
)  
)```
C.3 Linux Bash Scripts

Contents of batchme:

```bash
# Created by David Cerantola May 2014
# Purpose: Execute this script to generate a mesh and run background parallel 2D Fluent
# Output: converged cfd solution outputf1.cas/outputf1.dat and results file Yflresultsf.txt
#
# To run on linux computer: bash batchme

#VARIABLES
#echo "Enter Working directory:" 
#read rootdir 
rootdir=/home/mcl222/David/test

#ICEM grid journal file
mesh=coarsegrid2D
#Fluent journal file
fljou=firke2DS0

#echo "Enter Icore spawn:" 
#read hnum
hnum=8
#requires file hostfile$hnum. For four nodes on icore 8, uncomment next two lines
# cd ${rootdir}
# printf 'icore8
icore8
icore8
icore8' > hostfile8

# Working folder
for childname in a 
do

## ICEM ##
### COMMENT IF HAVE GRID
cd ${rootdir}
icemcfd -batch  -script ${rootdir}/${mesh}.tcl
rm project.*
rm hex.uns
echo "grids good"
### END IF HAVE GRID COMMENT

mkdir -p ${rootdir}/${childname}

## FLUENT ##
#command runs 2D double-precision Fluent on 4 nodes in background mode without # the GUI
/ansys_inc/v150/fluent/bin/fluent 2ddp -t4 -cnf=${rootdir}/hostfile$hnum -g -ssh < ${rootdir}/${fljou}.jou > & outputfile &
```
C.4. MESH GENERATION SCRIPT

To successfully execute this script, the word wraps must be manually removed and vertex numbers must agree with those shown in Fig. C.1. If compiling on a Windows machine, interchange the ‘#’ signs in the ‘ic_exec’ commands near the end of the script.

Text for coarsegrid2D.tcl:

```tcl
# ICEM CFD Mesh Generator
```
Figure C.1: Config. bb coarse grid generated from included script

# Created by David Cerantola May 2014
# Program designed such that should only need to modify spaces and file location
# dx = max. axial distance, dy = max. height distance
# spc1 = first space on outer wall and annulus up to start of CB
# spc2 = first space at CB trailing edge
# Input: Formatted point file Wallpts.dat consisting of 8 rows tab-delimited
# (x,y,z)
# Loaded points
# D=pnt0 E=pnt1 F=pnt2 G=pnt3 H=pnt4
# S=pnt5 T=pnt6 U=pnt7
# Output to Fluent: mesh2Dc.msh
##############################################################################
#load point data files for 2D mesh
ic_geo_cre_geom_input Wallpts.dat 1e-8 input PNTS pnt CRV S {} SURFS {}

#place point at origin
ic_point () PNTS pntOrig 0,0,0
set rD [ic_geo_pnt_dist pntOrig pnt0]
set rU [ic_geo_pnt_dist pnt4 pnt7]

#flange thickness for pt.S and pt.T
set flthk 0.003264
#lthroat = length to location of CS t at x=-0.23Do
set lthroat 0.03448

#### MESH VARIABLES ####

# average axial space
set dx 0.0022
# average height space
set dy [expr {$dx*0.5}]
# boundary layer spaces
set spc1 [expr {0.9*$dy}]
set spc2 [expr {0.9*$dy}]

# lbr = axial last cell length on bridge (pt.D).
set lbr [expr {0.6*$dx}]
# le = axial last cell length in diffuser (pt.H)
set le [expr {0.6*$dx}]

# boundary layer height modifications to give more uniform yplus
set blmodhin1 1
set blmodhout1 1
set blmodhin3 0.5
set blmodhout3 1
set blmodhout3b 1.5
set blmoddiffin 1.2
set blmodhexit 1.1

set ax1mod 1.25

# geometric growth ratios
set rathi 1.05
set rath 1.05
set rathptS 1.08
set rathcb 1.1
set ratax2 1.1
set ratax4 1.1

set rath2b 1.1
set rathe 1.1
set rathea 1.01
### END VARIABLES ###

### CREATE GEOMETRY ###

# points for CS t upstream of plane DS
ic_point {} PNTS pntDup pnt0-vector($lthroat,0,0)
ic_point {} PNTS pntSup pnt5-vector($lthroat,0,0)

# Flange points at T
ic_point {} PNTS pntSb pnt5+vector($flthk,0,0)
ic_point {} PNTS pntTa pnt6-vector($flthk,0,0)
ic_point {} PNTS pntTb pnt6+vector($flthk,0,0)

# symmetry plane curves
# inlet
ic_curve point CRVS crv1 (pntDup pntSup)
# CB
ic_curve point CRVS crv2 (pntDup pnt0)
ic_curve point CRVS crv3a (pnt0 pnt1)
ic_curve point CRVS crv3b (pnt1 pnt2)
ic_curve point CRVS crv3c (pnt2 pnt3)
ic_geo_cre_crv_concat CRVS crv3 1e-8 {crv3a crv3b crv3c}
#axis
ic_curve point CRVS crv4 (pnt3 pnt4)
#OW
ic_curve point CRVS crv5 (pntSup pnt5)
ic_curve point CRVS crv6a (pnt5 pntSb)
ic_curve point CRVS crv6b (pntSb pntTa)
ic_curve point CRVS crv6c (pntTa pntTb)
ic_curve point CRVS crv6d (pntTb pnt7)
ic_geo_cre_crv_concat CRVS crv6 1e-8 {crv6a crv6b crv6c crv6d}
#outlet
ic_curve point CRVS crv7 (pnt0 pnt5)
ic_curve point CRVS crv8 (pnt4 pnt7)

#Delete unused points and curves
ic_delete_geometry curve names {crv3a crv3b crv3c crv6a crv6b crv6c crv6d} 0 1

# define part names for boundary condition curves
ic_geo_set_part curve crv1 INLET 0
ic_geo_set_part curve crv2 WALL_INA 0
ic_geo_set_part curve crv3 WALL_IN 0
ic_geo_set_part curve crv4 CL 0
ic_geo_set_part curve crv5 WALL_OUTA 0
ic_geo_set_part curve crv6 WALL_OUT 0
ic_geo_set_part curve crv7 THROAT 0
ic_geo_set_part curve crv8 OUTLET 0

##############################################################################
#### BLOCKING ######
#Volume inside diffuser is called FLUID
ic_geo_new_family FLUID
#2D Planar
ic_hex_initialize_mesh 2d new_numbering new_blocking FLUID
#block splits
ic_hex_split_grid 11 19 pntOrig m PNTS CRVS INLET THROAT OUTLET WALL_INA WALL_IN
WALL_OUTA WALL_OUT CL FLUID VORFN
ic_hex_split_grid 33 19 pnt3 m PNTS CRVS INLET THROAT OUTLET WALL_INA WALL_IN
WALL_OUTA WALL_OUT CL FLUID VORFN
ic_hex_split_grid 11 13 pntDup m PNTS CRVS INLET THROAT OUTLET WALL_INA WALL_IN
WALL_OUTA WALL_OUT CL FLUID VORFN

#delete blocks inside CB
ic_hex_mark_blocks unmark
ic_hex_mark_blocks superblock 4
ic_hex_mark_blocks superblock 10
ic_hex_change_element_id VORFN
ic_delete_empty_parts

#Move vertices to corresponding points
ic_hex_move_node 41 pntDup
ic_hex_move_node 42 pnt0
ic_hex_move_node 43 pnt2
ic_hex_move_node 37 pnt3
ic_hex_move_node 19 pnt4
ic_hex_move_node 13 pntSup
ic_hex_move_node 34 pnt5
ic_hex_move_node 21 pnt7

#Associate edges to curves
ic_hex_set_edge_projection 41 13 0 1 crv1
ic_hex_project_to_surface 41 13
ic_hex_set_edge_projection 41 42 0 1 crv2
ic_hex_project_to_surface 41 42
ic_hex_set_edge_projection 42 43 0 1 crv3
ic_hex_project_to_surface 42 43
ic_hex_set_edge_projection 42 43 0 1 crv3
ic_hex_project_to_surface 42 43
ic_hex_set_edge_projection 37 43 0 1 crv3
ic_hex_project_to_surface 37 43
ic_hex_set_edge_projection 37 19 0 1 crv4
ic_hex_project_to_surface 37 19
ic_hex_set_edge_projection 13 34 0 1 crv5
ic_hex_project_to_surface 13 34
ic_hex_set_edge_projection 34 38 0 1 crv6
ic_hex_project_to_surface 34 38
ic_hex_set_edge_projection 38 21 0 1 crv6
ic_hex_project_to_surface 38 21
ic_hex_set_edge_projection 42 34 0 1 crv7
ic_hex_project_to_surface 42 34
ic_hex_set_edge_projection 19 44 0 1 crv8
ic_hex_project_to_surface 19 44
ic_hex_set_edge_projection 44 21 0 1 crv8
ic_hex_project_to_surface 44 21

#additional splits
# split on panel FG is to give representative core block if meshed as 3D
ic_hex_split_grid 37 43 0.3 m PNTS INLET THROAT OUTLET WALL_INA WALL_IN
WALL_OUTA WALL_OUT CL FLUID VORFN
# split to dRt not really necessary but gives control to bias cell density in height
ic_hex_split_grid 41 13 0.5 m PNTS INLET THROAT OUTLET WALL_INA WALL_IN
WALL_OUTA WALL_OUT CL FLUID VORFN

# better values for intermediate vertices on outlet
ic_hex_get_node_location {49} _tempx _tempy _tempz
ic_hex_set_node_location y [expr {$_tempy} -csys global node_numbers {50}]
ic_hex_get_node_location {43} _tempx _tempy _tempz
ic_hex_set_node_location y [expr {$_tempy} -csys global node_numbers {44}]

set lET [expr [ic_hex_get_edge_param 43 55 len] + [ic_hex_get_edge_param 55 38 len]]
ic_hex_set_node_location x [expr {$_tempx + $1ET*cos(1.31)}] y [expr {$_tempy +}
APPENDIX C. CFD EXECUTABLE SCRIPTS

```bash
$lET*\sin(1.31)} -csys global node_numbers {38}
ic_hex_set_node_location x [expr \($\_\text{tempx} + 0.5*\$lET*\cos(1.31)}) \ y [expr \($\_\text{tempy} + 0.5*\$lET*\sin(1.31)}) -csys global node_numbers {55}
ic_hex_get_node_location {55} _\text{tempx} _\text{tempy} _\text{tempz}
ic_hex_set_node_location y [expr _\text{tempy}] -csys global node_numbers {56}

#To get different body for Tduct
ic_geo_new_family TDUCT
ic_hex_mark_blocks unmark
ic_hex_mark_blocks superblock 16
ic_hex_mark_blocks superblock 26
ic_hex_change_element_id TDUCT
#End different bodies

#resnap project vertices
ic_hex_project_to_surface PNTS WALL_INA WALL_IN WALL_OUTA WALL_OUT INLET OUTLET THROAT CL FLUID TDUCT

#### END BLOCKING ######

##### EDGE MESHING ######

#Edge Meshing parameters (in blocking)

#Axial length Ax1 in tduct sax1=lthroat
#Geometric2 mesh law
set sax1 [ic_hex_get_edge_param 41 42 len]
set ratax1 [expr {($sax1-$lbr)/($sax1-$ax1mod* $dx)}]
set numax1 [ceil [expr {log(1-$sax1/($lbr) * (1-$ratax1))/log($ratax1)+1}]]
message displays on GUI
mess "numax1 $numax1

#to maintain correct x and y spacing since vectors angled
at pt.E
ic_hex_get_node_location {38} _\text{tempx} _\text{tempy} _\text{tempz}
ic_point {} PNTS pnt38 [expr _\text{tempx}],[expr _\text{tempy}],[expr _\text{tempz}]
ic_hex_get_node_location {44} _\text{tempx} _\text{tempy} _\text{tempz}
ic_point {} PNTS pnt44 [expr _\text{tempx}],[expr _\text{tempy}],[expr _\text{tempz}]
set vec1 [ic_geo_vec_nrm [ic_geo_vec_diff pnt2 pnt3]]
set vec2 [ic_geo_vec_nrm [ic_geo_vec_diff pnt2 pnt44]]
set vec3 [ic_geo_vec_nrm [ic_geo_vec_diff pnt2 pnt38]]
set dotprod1 [expr acos([ic_geo_vec_dot [expr vec1] [expr vec2] ])]
set dotprod2 [expr acos([ic_geo_vec_dot [expr vec1] [expr vec3] ])]

#at pt.D and pt.S
set vec4 [ic_geo_vec_nrm [ic_geo_vec_diff pnt0 pntDup]]
set vec5 [ic_geo_vec_nrm [ic_geo_vec_diff pnt0 pnt1]]
set dotprod4 [expr acos([ic_geo_vec_dot [expr vec4] [expr vec5] ])]
set vec6 [ic_geo_vec_nrm [ic_geo_vec_diff pnt5 pntSup]]
set vec7 [ic_geo_vec_nrm [ic_geo_vec_diff pnt5 pnt6]]
set dotprod6 [expr acos([ic_geo_vec_dot [expr vec6] [expr vec7] ])]
```

C.4. MESH GENERATION SCRIPT

ic_hex_set_mesh 41 42 n $numax1 h1 [expr {$ax1mod*$dx}] h2 [expr {$lbr}] r1 1 r2 [expr {$ratax1}] lmax [expr {$ax1mod*$dx}] default unlocked
ic_hex_set_mesh 53 54 n $numax1 h1 [expr {$ax1mod*$dx}] h2 [expr {$lbr}] r1 1 r2 $ratax1 lmax [expr {$ax1mod*$dx}] default unlocked
ic_hex_set_mesh 13 34 n $numax1 h1 [expr {$ax1mod*$dx}] h2 [expr {$lbr}] r1 1 r2 $ratax1 lmax [expr {$ax1mod*$dx}] default unlocked
#end Ax1

#Ax2 diffuser inlet pnt. D to pnt. F 0.9* a fudge factor.
#big geometric mesh law
set sax2 [ic_hex_get_edge_param 42 43 len]
set rx2a [expr {log($dx/($lbr/abs(cos($dotprod4))))/log($ratax2)}]
#1st guess
set numax2a [expr {0.9 * $sax2/(4 * 0.5 * $dx)}]
for {set inc 0} {$inc<10} {incr inc} {set numax2a [expr {0.9 * 4 * $sax2/(((1+3 * pow($ratax2,($rx2a/($numax2a*$numax2a+1))))+3 * pow($ratax2,($rx2a/($numax2a*$numax2a+1)))+$dx/($lbr/abs(cos($dotprod4)))))/2}]

set rx2b [expr {log($dx/($spc2/sin($dotprod1))))/log($ratax4)}]
set numax2b [expr {0.9 * $sax2/(4 * 0.5 * $dx)}]
for {set inc 0} {$inc<10} {incr inc} {set numax2b [expr {0.9 * 4 * $sax2/(((1+3 * pow($ratax4,($rx2b/($numax2b*$numax2b+1))))+3 * pow($ratax4,($rx2b/($numax2b*$numax2b+1)))+$dx/($spc2/sin($dotprod1))))/2}]
set numax2c [ceil [expr {2 * $numax2a+2 * $numax2b-4}]]

ic_hex_set_mesh 42 43 n $numax2 h1 [expr {$lbr/abs(cos($dotprod1))}] h2 [expr {$spc2/sin($dotprod1)}] r1 $ratax2 r2 $ratax2 lmax [expr {$dx}] default unlocked
ic_hex_set_mesh 54 55 n $numax2 h1 [expr {$lbr/abs(cos($dotprod6))}] h2 [expr {$spc2/sin($dotprod1)}] r1 $ratax2 r2 $ratax2 lmax [expr {$dx}] default unlocked
ic_hex_set_mesh 34 38 n $numax2 h1 [expr {$lbr/abs(cos($dotprod6))}] h2 [expr {$spc2/sin($dotprod1)}] r1 $ratax2 r2 $ratax2 lmax [expr {$dx}] default unlocked
# end Ax2

# height, originating from annulus
set sh [expr {{ic_hex_get_edge_param 42 54 len} + {ic_hex_get_edge_param 54 34 len}}]
set rx [expr {ds1*rat1*(2*(N^2+N)/2)} where N ~ N/4 so want N=4N-2+1]
# numh a guess first, then iterates, then corrects for entire length
set numh [expr {$sh/(2 * $dy)}]
for {set inc 0} {$inc<10} {incr inc} {set numh [expr {4 * $sh/(($spc1) + 1+3 * pow($rathi,($rx/($numh+1))))+3 * pow($rathi,($rx/($numh+1)))+$dy/($spc1))/2}]
set numh [ceil [expr {4 * $numh-2}]]
set frha [expr {{ic_hex_get_edge_param 42 54 len}/$sh}]
set numha [ceil [expr {1.1 * $frha * $numh +2}]]
set numhb [ceil [expr {1.1 * (1-$frha) * $numh}]]
mess "numha $numha numhb $numhb
"
APPENDIX C. CFD EXECUTABLE SCRIPTS

#height, o-grid centre (at centreline)
set sh2 [ic_hex_get_edge_param 37 49 len]
set numh2 [ceil [expr {$sh2/$dy+1}]]
mess "numh2 $numh2\n"

#h2b - buffer region above central o-grid, originating off CB
set sh2b [ic_hex_get_edge_param 49 43 len]
set numh2b [ceil [expr {$sh2b/(2*$dy)+2}]]
mess "numh2b $numh2b\n"

set h2mod2 [expr {($numh2-1)/($numh2+$numh2b +($numha + 0.9*$numhb)-4.0)}]
set h2mod240 [expr {($numh2+$numh2b-2)/($numh2+$numh2b +($numha + 0.9*$numhb)-4.0)}]
set h2mod597 [expr {($numh2+$numh2b+$numha-3)/($numh2+$numh2b+($numha + 0.95*$numhb)-4.0)}]

#outlet vertices moved so uniform height in core
ic_hex_set_node_location y [expr {1.0*$h2mod2 * $rU}] -csys global node_numbers {50}
ic_hex_set_node_location y [expr {1.0*$h2mod240 * $rU}] -csys global node_numbers {44}
ic_hex_set_node_location y [expr {1.0*$h2mod597 * $rU}] -csys global node_numbers {56}

#uniform outlet cell height = lh2e
set sh2e2 [ic_hex_get_edge_param 19 50 len]
set lh2e [expr {($sh2e2)/($numh2-1)}]

#h2 - central o-grid
ic_hex_set_mesh 37 49 n $numh2 h1 $dy h2 $dy r1 $rath2b r2 $rath2b lmax $dx
default unlocked
ic_hex_set_mesh 19 50 n $numh2 h1 $lh2e h2 $lh2e r1 $rath2b r2 $rath2b lmax $dx
default unlocked

#h2b - buffer region above central o-grid, originating off CB
ic_hex_set_mesh 49 43 n $numh2b h1 [expr {$lh2e}] h2 [expr {($spc2)}] r1 $rath2b r2 $rath2b lmax $dy default unlocked
ic_hex_set_mesh 50 44 n $numh2b h1 $lh2e h2 $lh2e r1 $rath2b r2 $rath2b lmax $lh2e default unlocked

#height, originating from throat inlet
ic_hex_set_mesh 41 53 n $numha h1 [expr {($b1modhin1*$spc2)}] h2 [expr {1.5*$dy}] r1 $rathi r2 1 lmax [expr {1.5*$dy}] default unlocked
ic_hex_set_mesh 53 13 n $numhb h1 [expr {1.5*$dy}] h2 [expr {($b1modhout1*$spc1)}] r1 1 r2 $rathi lmax [expr {1.5*$dy}] default unlocked
ic_hex_set_mesh 42 54 n $numha h1 [expr {($spc2*$b1modhin3/abs(cos($dotprod4))})] h2 [expr {($dy)}] r1 $rath 2 1 lmax $dy default unlocked
ic_hex_set_mesh 54 34 n $numhb h1 [expr {($dy)}] h2 [expr {($spc1*$b1modhout1/abs(cos($dotprod6))})] r1 1 r2 $rath 1 lmax $dy default unlocked
ic_hex_set_mesh 43 55 n $numha h1 [expr {($dy)}] h2 [expr {1.5*$dy/sin($dotprod2)}] r1 $rathcb r2 $rath 2 $rath lmax [expr {1.4*$dx}] default unlocked
ic_hex_set_mesh 55 38 n $numhb h1 [expr {2*$dy/sin($dotprod2)}] h2 [expr {($spc1*1.2/$b1modeffin1)}] r1 $rathcb r2 $rathpt $lmax [expr {1.4*$dx}] default unlocked
ic_hex_set_mesh 44 56 n $numha h1 [expr {1*$lh2e}] h2 [expr {0.9*$lh2e}] r1 1 r2
#Ax4 in diffuser, 0.9* fudge
set sax4 [ic_hex_get_edge_param 43 44 len]
set rx4b [expr {log($dx/($le))/log($ratax4)}]
set numax4b [expr {0.9* $sax4/(4 * 0.5 * $dx)}]
for {set inc 0} {$inc<10} {incr inc} {set numax4b [expr {(0.9 * 4 * $sax4/($le) *(1+3*pow($ratax4,($rx4b/($numax4b/3+1))))+3* pow($ratax4,($rx4b/($numax4b/3+1))))*(1+3*pow($ratax4,($rx4b/($numax4b/3+1))))+3* pow($ratax4,($rx4b/($numax4b/3+1))))+1)/2]}
set numax4 [ceil [expr {2 * $numax4a+2 * $numax4b-4}]]
mess "numax4 $numax4\n"
# default better for plug, biexp better for conical CB
ic_hex_set_mesh 37 19 n $numax4 h1 [expr {0.6*$$pc2/sin($dotprod1)}] h2 [expr
($pc2/$sin($dotprod1)) r1 1 r2 [expr {$ratax4}] lmax $dx default unlocked
#end Ax4
#end meshing parameters

#Edit Edges for better fit to curves
#ax3 edit edge
ic_hex_undo_major_start auto_edge_split
ic_hex_auto_split_edge 42 43
ic_hex_auto_split_edge 38 21
ic_hex_undo_major_end auto_edge_split
#end edit edges

#match edge spacing between adjacent blocks
#height from tduct
ic_hex_match_edges 41 53 53 13
ic_hex_match_edges 42 54 54 34
ic_hex_match_edges 43 55 55 38
ic_hex_match_edges 37 49 49 43
#outlet
ic_hex_match_edges 19 50 50 44
ic_hex_match_edges 50 44 44 56
ic_hex_match_edges 44 56 56 21
### END MESHING ###

### OUTPUT ###

#Create multi block mesh and load mesh from blocking
ic_hex_create_mesh WALL_INA WALL_IN WALL_OUTA WALL_OUT INLET OUTLET THROAT CL FLUID TDUCT proj 2 dim_to_mesh 3

ic_hex_write_file hex.uns WALL_INA WALL_IN WALL_OUTA WALL_OUT INLET OUTLET THROAT CL FLUID TDUCT proj 2 dim_to_mesh 3 -family_boco family_boco.fbc
#file maxdim quiet reset_family_prefix check_orient
ic_uns_load hex.uns 3 0 {} 2
ic_empty_boco
ic_boco_nastran_csystem reset

#Output stuff
ic_boco_solver {ANSYS Fluent}
ic_solver_mesh_info {ANSYS Fluent}

#boundary conditions
ic_boco_set CL {{1 AXIS 0}}
ic_boco_set INLET {{1 (MASFI) 0}}
ic_boco_set OUTLET {{1 PRESO 0}}
ic_boco_set THROAT {{1 INTER 0}}
ic_boco_set WALL_IN {{1 WALL 0}}
ic_boco_set WALL_OUT {{1 WALL 0}}
ic_boco_set WALL_INA {{1 WALL 0}}
ic_boco_set WALL_OUTA {{1 WALL 0}}
ic_boco_set WALL_IN {{1 WALL 0}}
ic_boco_set WALL_OUT {{1 WALL 0}}
ic_boco_set WALL_INA {{1 WALL 0}}
ic_boco_set WALL_OUTA {{1 WALL 0}}
ic_boco_set FLUID {{1 FLUID 0}}
ic_boco_set TDUCT {{1 FLUID 0}}

#Output control
### replace below with for bashing
ic_boco_save project.fbc
ic_boco_save_atr project.atr
mess "waiting for mesh output\n"

#For Windows
ic_exec {C:/Program Files/ANSYS Inc/v150/icemcfd/win64_amd/icemcfd/output-interfaces/fluent6} -dom hex.uns -b project.fbc -dim2d mesh2Dc
#For linux
ic_exec {/ansys_inc/v150/icemcfd/linux64_amd/icemcfd/output-interfaces/fluent6} -dom hex.uns -b project.fbc -dim2d mesh2Dc
mess "mesh made\n"
exit
C.5  CFD Journal Script

If running the script directly from a working directory as opposed to bashing batchme, remove the `../` in front of mesh2Dc.msh and BCinlet.prof.

Contents of flrke2DS0.jou:

```
; Created by David Cerantola May 2014
; Purpose: Journal for running Fluent to converge a diffuser geometry solution
; Inputs: mesh2Dc.msh and BCinlet.prof (one directory up)
; Output: converged rke solution outputfl1.cas/.dat and post-processed file
;       Yflresultsf.txt
;
/file/start-transcript transcript.trn

; mesh is one directory back
/file/read-case ../mesh2Dc.msh

; in batch mode 1.confirm file overwrite 2.exit on error 3.hide questions
/file/set-batch-options yes yes no

;2D axisymmetric swirl
.define/models/axisymmetric? yes
.define/models/swirl? yes

;;;;;;;;;;;;;;;;;;;;;;;
; Models and materials 
;;;;;;;;;;;;;;;;;;;;;;;

; energy model questions: 1. enable 2. viscous energy dissipation -yes 3. inlet
diffusion
.define/models/energy? yes yes yes

; turbulence model
; for ke-realizable 1. enable
.define/models/viscous/ke-realizable? yes

; for non-eq 1. enable
.define/models/viscous/near-wall-treatment/non-equilibrium-wall-fn? yes

;;; Constant density to start
;material questions 1. density 2. Cp 3. thermal conductivity 4. viscosity 5. MW 
6. thermal expansion 7. speed of sound
.define/materials/change-create air air yes constant 1.225 yes
  piecewise-polynomial 2 100. 1000. 8 1161.48214452351 -2.36881890191577
  0.0148551108358867 -5.03490927522584e-05 9.9285695564e-08
  -1.1109658897742e-10 6.54019600406048e-14 -1.57358768447275e-17 1000. 3000.
  8 -7069.81410143802 33.7060506468204 -0.05812759533375815 5.4216153229608e-05
  -2.93678858119e-08 9.237533169567681e-12 -1.56555339604519e-15
  1.11233485020759e-19 yes piecewise-linear 2 300 0.0263 800 0.0573 yes sutherland
  three-coefficient-method 1.716e-05 273.11 110.56 no no no
```
;;; Read inlet profile in one directory back
/file/read-profile ../BCinlet.prof

;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;
;; Boundary Conditions ;
;; mass flow inlet no swirl
/define/boundary-conditions/mass-flow-inlet inlet yes no yes no "inletbc"
"mass-flux-act" no 302 no 0 yes no 1 no 0 no no yes 5 0.07194
/report/reference-values/compute mass-flow-inlet inlet

;;; pressure outlet p=0g, T0=300K, dir=(1,0,0), TI=5%
/define/boundary-conditions/pressure-outlet outlet no 0. no 299. no yes no yes 1
0.0004 no no

; plenum not modelled in mesh
; plenum_in, plenum_top p=10g, T0=300K, dir=(1,0,0), TI=5%
/define/boundary-conditions/pressure-inlet plenum_in yes no 0. no 0. no 299. no
yes no 1 0.0004
/define/boundary-conditions/pressure-inlet plenum_top yes no 0. no 0. no 299. no
yes no 1 0.0004

; Solution methods
; SIMPLE=20, SIMPLEC=21
/solve/set/p-v-coupling 21
;SIMPLEC skewness=1
/solve/set/p-v-controls 1

; Solution controls
; limits pmin=1000a pmax=1e7a Tmin=100K Tmax=1000K visc ratio=1e7
/solve/set/limits 1000 1000000. 100 1000 1e-14 1e-20 10000000.
; limiter 0=std, 1=multidim 2=differentiable. no=cell-to-cell, yes=cell-to-face
/solve/set/slope-limiter-set/2 yes no
; Monitors 1.continuity 2.x-vel 3.y-vel 4.z-vel 5.energy 6.k 7.epsilon
/solve/monitors/residual/convergence-criteria 1e-03 1e-03 1e-03 1e-03 1e-06
1e-03

; surface monitor 1.plot 2.print 3.write file
/solve/monitors/surface/set-monitor pbart "Area-Weighted Average" pressure inlet
() no yes no 10
/solve/monitors/surface/set-monitor pdynbart "Mass-Weighted Average" dynamic-pressure inlet () no yes no 10
/solve/monitors/surface/set-monitor Ume "Mass-Weighted Average" axial-velocity outlet () no yes no 10
/solve/monitors/surface/set-monitor tau_win "Area-Weighted Average" axial-wall-shear wall_in () no yes no 10

; Initialization
/solve/initialize/set-defaults/temperature 302
/solve/initialize/set-defaults/x-velocity 50
/solve/initialize/set-defaults/y-velocity 0
/solve/initialize/set-defaults/k 5
/solve/initialize/set-defaults/epsilon 400
/solve/initialize/initialize-flow

; Auto-save
/file/auto-save/root-name output
/file/auto-save/data-frequency 100
/file/auto-save/case-frequency if-mesh-is-modified
/file/auto-save/retain-most-recent-files yes
/file/auto-save/max-files 2

/solve/set/reporting-interval 10

; under-relaxation factors reduced to avoid sudden changes due to imbalance of
; fluxes after initialization/patching
/solve/set/under-relaxation/pressure 0.2
/solve/set/under-relaxation/density 0.6
/solve/set/under-relaxation/body-force 0.6
/solve/set/under-relaxation/mom 0.4
/solve/set/under-relaxation/w-swirl 0.4
/solve/set/under-relaxation/k 0.4
/solve/set/under-relaxation/epsilon 0.4
/solve/set/under-relaxation/omega 0.4
/solve/set/under-relaxation/turb-viscosity 0.4
/solve/set/under-relaxation/temperature 0.9

/solve/iterate 20

; pressure schemes 10=standard 14=presto 11=linear 12=second order 13=body force
; other schemes 0= first order upwind 1=second order upwind 2=power law
; 3=central differencing 4=quick 6=third order muscl 7=bounded central diff
/solve/set/discretization-scheme/pressure 10
/solve/set/discretization-scheme/density 0
/solve/set/discretization-scheme/mom 0
/solve/set/discretization-scheme/w-swirl 0
/solve/set/discretization-scheme/k 0
/solve/set/discretization-scheme/epsilon 0
/solve/set/discretization-scheme/omega 0
/solve/set/discretization-scheme/temperature 1

/solve/set/under-relaxation/pressure 0.7
/solve/set/under-relaxation/density 1
/solve/set/under-relaxation/body-force 1
/solve/set/under-relaxation/mom 0.7
/solve/set/under-relaxation/w-swirl 0.9
/solve/set/under-relaxation/k 0.9
/solve/set/under-relaxation/epsilon 0.9
/solve/set/under-relaxation/omega 0.9
/solve/set/under-relaxation/turb-viscosity 1
/solve/set/under-relaxation/temperature 1

/solve/iterate 1000

;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;
;;;; Implement Ideal Gas ;;;;;
;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;
; material questions 1. density 2. Cp 3. thermal conductivity 4. viscosity 5. MW
6. thermal expansion 7. speed of sound

/define/materials/change-create air air yes ideal-gas yes piecewise-polynomial
2 100. 1000. 8 1161.48214552351 -2.36881890191577 0.01485 51108358867
-5.03490927522584e-05 9.9285695564579e-08 -1.1109658897742e-10
6.540196040604e-14 -1.57358768447275e-17 1000. 3000. 8 -7069.81410143802
33.706506468204 -0.0581275953375815 5.42161532229608e-05 -2.936678858119e-08
9.237531169567681e-12 -1.5655339604519e-15 1.1123348502759e-19 yes
piecewise-linear 2 300 0.0263 800 0.0573 yes sutherland three-coefficient-method
1.716e-05 273.11 110.56 no no no

; change scaling to local (ie rms residuals) 1.scale residuals 2.compute local
; scale 3.enable local scaling
/solve/set/under-relaxation/pressure 0.2
/solve/set/under-relaxation/density 0.6
/solve/set/under-relaxation/body-force 0.6
/solve/set/under-relaxation/mom 0.4
/solve/set/under-relaxation/w-swirl 0.4
/solve/set/under-relaxation/k 0.4
/solve/set/under-relaxation/epsilon 0.4
/solve/set/under-relaxation/omega 0.4
/solve/set/under-relaxation/turb-viscosity 0.4
/solve/set/under-relaxation/temperature 0.6

; pressure schemes 10=standard 14=presto 11=linear 12=second order 13=body force
; other schemes 0= first order upwind 1=second order upwind 2=power law
; 3=central differencing 4=quick 6=third order muscl 7=bouned central diff
/solve/set/discretization-scheme/pressure 12
/solve/set/discretization-scheme/density 4
/solve/set/discretization-scheme/mom 4
/solve/set/discretization-scheme/w-swirl 4
/solve/set/discretization-scheme/k 4
/solve/set/discretization-scheme/epsilon 4
/solve/set/discretization-scheme/omega 4
/solve/set/discretization-scheme/temperature 4

/solve/iterate 100

/solve/set/under-relaxation/pressure 0.5
/solve/set/under-relaxation/body-force 1
/solve/set/under-relaxation/temperature 0.8
/solve/set/under-relaxation/omega 0.8

C.6   FORTRAN POST-PROCESSING SCRIPT

Contents of fortransorter.f90:

!Program by David Cerantola May 2014
!Input: resultsXX.txt XX =01--10
!NCHILD number of files
!NVAR number of variables in file [pbar qmf p0mf ubar umf ]
!NPLANE number of cross sections evaluated (inlet, outlet)
!NOBJ number of objectives
!
!Purpose: nicely sorts/outputs data from Fluent Surface Integral Report
!Yflresultsf.txt into a readable array and creates an objective results
!file obj.txt
!
/solve/set/under-relaxation/turb-viscosity 1
/solve/set/under-relaxation/temperature 0.96
/solve/set/discretization-scheme/pressure 12
/solve/set/discretization-scheme/density 4
/solve/set/discretization-scheme/mom 4
/solve/set/discretization-scheme/w-swirl 4
/solve/set/discretization-scheme/k 4
/solve/set/discretization-scheme/epsilon 4
/solve/set/discretization-scheme/omega 4
/solve/set/discretization-scheme/temperature 4
/file/write-case output-1b
/solve/iterate 8000
/file/write-case-data outputf1
/report/surface-integrals/area-weighted-avg inlet outlet () pressure y Yflresultsf.txt
/report/surface-integrals/mass-weighted-avg inlet outlet () dynamic-pressure y Yflresultsf.txt y
/report/surface-integrals/mass-weighted-avg inlet outlet () total-pressure y Yflresultsf.txt y
/report/surface-integrals/area-weighted-avg inlet outlet () axial-velocity y Yflresultsf.txt y
/report/surface-integrals/mass-weighted-avg inlet outlet () axial-velocity y Yflresultsf.txt y
/file/stop-transcript
/exit
yes

C.6   Fortran Post-Processing Script

Contents of fortransorter.f90:

!Program by David Cerantola May 2014
!Input: resultsXX.txt XX =01--10
!NCHILD number of files
!NVAR number of variables in file [pbar qmf p0mf ubar umf ]
!NPLANE number of cross sections evaluated (inlet, outlet)
!NOBJ number of objectives
!
!Purpose: nicely sorts/outputs data from Fluent Surface Integral Report
!Yflresultsf.txt into a readable array and creates an objective results
!file obj.txt
!
Compilation:

```
! Compile on linux fortran g95 with
! g95 -o <output executable> <input.f90>
! Run program in current directory by: ./<output executable> ; or:
!/home/mcl222/David/test/fortsort
! g95 -o fortsort fortransorter.f90
!Output: propXX.txt
! obj.txt = [Cb gamma Cp0]
!
! Notes: if changing file names, also need to correct number of characters
```

Program:

```
PROGRAM FLUENTRESULTS

!User Inputs
INTEGER, PARAMETER :: NCHILD = 1
INTEGER, PARAMETER :: NVAR = 5
INTEGER, PARAMETER :: NPLANE = 2
INTEGER, PARAMETER :: NOBJ = 3

!Variable Identifiers
INTEGER :: n = 0
INTEGER :: i,j = 0
INTEGER :: gen = 0

!Array(columns,rows)
REAL*8 :: prop(NPLANE,NVAR) = 0
REAL*8 :: obj(3,NCHILD) = 0

CHARACTER*30 :: rootdir = "/home/mcl222/David/test/objout"
CHARACTER*8 propformat
CHARACTER*9 objformat
CHARACTER*50 filein
CHARACTER*49 fileout
CHARACTER*46 fileobj
CHARACTER*12 HEADER

!!!!!!!!!!!!!!
!PROGRAM BODY!
!!!!!!!!!!!!!!

!!!For interactive input
!WRITE(*,*) 'Enter Todays date (0522):'
!READ(*,*) date

!Format of read/write variables
WRITE (propformat, '(a1,i1,a6)') '(', NPLANE,'f16.8)'
WRITE (objformat, '(a1,i2,a6)') '(', 3,'f16.8)'

!f16.0 - real number, 16 digits with no implied decimal places
100 format(42x,f16.0)

!Get results fileout
DO n =1, NCHILD
!files must be in working directory
```
if (n < 10) then
    WRITE (fileout, "/(a,A,I1,A4)") , rootdir, "/prop0", n, ".txt"
    WRITE (filein, "/(a,A,I1,A)") , rootdir, "/flres0", n, ".txt"
else
    WRITE (fileout, "/(a,A,I2,A)") , rootdir, "/prop", n, ".txt"
    WRITE (filein, "/(a,A,I2,A)") , rootdir, "/flres", n, ".txt"
endif

OPEN (UNIT = 20, FILE = filein, status="old", action="read", iostat=ierror)
if (ierror /= 0) then
    print*, "File not found"
    exit
else
    print*, "It worked"
end if

! Read in and Ignore Header
READ (20, *) HEADER
DO i = 1, NVAR
    READ (20, *) HEADER
    READ (20, *) HEADER
    READ (20, *) HEADER
    DO j = 1, NPLANE
        READ (20, 100) prop(j,i)
        !print*, prop(j,i)
    end do
    READ (20, *) HEADER
    READ (20, *) HEADER
end do

! matrix format = A(j,i) ie. (cols, rows)
obj = [Cb gamma Cp0]
obj(1,n) = -prop(1,1)/prop(1,2)
obj(2,n) = prop(2,4)/prop(2,5)
obj(3,n) = (prop(2,3) - prop(1,3))/prop(1,2)
!print*, obj
!pause

! Output formatted array of property data
OPEN (UNIT = 50, FILE = fileout, action="write")
WRITE (50, propformat) prop

CLOSE (UNIT = 20)
CLOSE (UNIT = 50)
end do

WRITE (fileobj, "/(a,A)") , rootdir, "/obj.txt"

OPEN (UNIT = 51, FILE = fileobj, action="write")
WRITE (51, objformat) obj
CLOSE (UNIT = 51)

END PROGRAM FLUENTRESULTS
Appendix D

Part Drawings

The component assembly drawings are shown Figs. D.1 and D.2 and reference the manufactured part drawings below.
**APPENDIX D. PART DRAWINGS**

---

**Figure D.1:** Transition duct assembly drawing

---

**Figure D.2:** Diffuser assembly drawing (Config. aa shown)
Figure D.3: Centre Body a

Figure D.4: Centre Body b
Figure D.5: Centre Body c

Figure D.6: Duct STa
Figure D.7: Duct STb

Figure D.8: Duct STc
Figure D.9: Duct TUA

Figure D.10: Duct TUb
Figure D.11: Duct TUc

Figure D.12: Duct SUd
Figure D.13: FlangeS

Figure D.14: FlangeTa
Figure D.15: FlangeTb

Figure D.16: Transition duct in cap
Figure D.17: Transition duct inner cone

Figure D.18: Transition duct end cap
Figure D.19: Tab
Appendix E

Diffuser Geometry Study Supplementary

This study evaluated the manufactured components selected from the optimization study completed in App. B with S0 swirl to determine the influence of geometry on performance. Through interchanging the centre-bodies (CBs a–c and no centre body, CBn) and outer walls (OWs a–c), 12 configurations were tested. Numerical simulations with experimental validation was the basis for [182].

The CFD for [182] evaluated 3D grids with 30° wedges whereas this study re-computed the simulations on 2D grids. On an absolute scale, discrepancies between the 2D and 3D grid results were within the numerical uncertainties; however, some deviations existed in the relative comparisons. These deviations did not affect the conclusions of the study.

E.1 Literature Review

In addition to the theory discussed in Ch. 2, the geometric shaping of the centre-body is an additional design variable for annular diffusers with fully open outer diameter outlets.

A centre-body (CB) is a device that brings termination to the inner cylinder of the upstream annular components and provides the exhaust system with an opportunity for area expansion into the core region. As a result, the flow path is prone to both diffusion and curvature characteristics. CB shapes ranging from an abrupt termination to a faired cone have been considered. Wood and Higginbotham [7] examined five diffusers of different lengths with varying CB profiles and recommended that the CB length should be roughly
one-half the overall diffuser length for an annulus with $D_i/D_o = 0.7$, and shorter diffusers with the same divergence angle resulted in reduced pressure recovery and a deterioration in exit velocity distribution. One of the CB shapes conformed to that recommended by Gibson [183] that was intended to produce uniform total pressure loss per unit length; however, Gibson’s study did not account for turbulence. Mallett and Harp’s [6] investigation of CB curvature revealed that the initially higher rate of expansion due to a flat panel suppressed separation whereas flow separated from the curved panel studies. Eckert et al. [8] considered the configurations shown in Fig. E.1 and found higher pressure recovery occurred with their long ellipsoidal CB that extended just beyond the outlet plane of the outer wall since it had a more gradual area variation than that provided by the short centre-body.

Figure E.1: Centre-bodies evaluated by Eckert et al. [8] and instrumentation locations

Adkins, Jacobsen and Chevalier [9] studied conical CBs in a diffuser with a constant diameter OW and near-uniform inlet unswirled flow. Pressure recovery increased as wall angle decreased (longer CBs were better for performance). Five cones were manufactured that had included angles of $10^\circ$ (truncated), $25^\circ$, $35^\circ$, $45^\circ$, and $60^\circ$. The cone with the $35^\circ$ angle showed the best performance.
45°, and 132°. In all cases, maximum pressure recovery occurred 4.0\(D_i\) downstream of the diffuser inlet. They chose to non-dimensionalize distances using \(CB\) diameter rather than annulus height since it directly correlated to the length of the \(CB\) wake.

Different area ratio ducts were tested in [9], and curvefits suggested that maximum pressure recovery occurred in diffusers with \(0.7 \lesssim D_i/D_o \lesssim 0.75\). To prevent flow separation at the cone/cylinder junction, radius of curvatures, \(R\), were added at the junction. For the 45° cone, \(R/D_i = 0.42\) provided a useful gain in pressure recovery but larger curvatures did not achieve further improvement. Three curved-wall CBs were also manufactured whose included angle was optimized using the correlations in [52] with respect to area ratio. The longest CB, \(L = 1.62D_i\), developed 14% more pressure recovery than the 25° cone (adding a blend radius offered no further improvements); however, the two shorter curved-wall CBs produced less recovery than the 35° cone but performance improved when a blend radius was applied at the junction.

### E.2 Experimental Results

Table E.1 shows the number of tests that were completed for each configuration with S0 swirl and no tabs. With the exception of Config. nc, all configurations were completed with at least three different mass flow rates. Configurations aa, bb, and cc \((CB|OW)\) were selected for in-depth investigation.

| Table E.1: Diffuser geometry study number of retained tests. |
|-----------------|---|---|---|---|
| OW \ CB | n  | a  | b  | c  |
| a      | 4  | 9  | 10 | 3  |
| b      | 4  | 7  | 15 | 3  |
| c      | 2  | 4  | 8  | 12 |

**E.2.1 Inlet**

Figure E.2 shows experimental pressure profiles obtained at the annular diffuser inlet \((section 1 \equiv x/D_o = -0.23)\) using Eq. (A.30) that were typical of all configurations tested without swirl. Pressure error bars were calculated uncertainties (95% CI) whereas the radial errors were attributed to completing the traverses manually (typical for all inlet pressure profiles). All profiles were similar with inner boundary layer thickness
of approximately $\delta_i = 5$ mm ($0.06R_o$). Velocity peaked nearer to the inner wall at $r \approx 0.84-0.87R_o$ and static pressure decreased gradually from inner to outer wall. The corresponding area blockages are given in Tab. E.2.

![Figure E.2: Diffuser geometry study Configs. aa, bb, and cc inlet pressure profiles. Symbols denote manometer readings.](image)

**Table E.2:** Diffuser geometry study inlet flow blockage, $A_{Bi}$. A chosen configuration was selected with $Re_t \approx 1.4 \times 10^5$ from all tests completed with the same configuration. The ranges provided in the ‘all data’ row were based on all tests with the given configuration over the range $0.9 \times 10^5 < Re_t < 3 \times 10^5$ and the ‘all’ column gives the overall average and range from all configurations.

<table>
<thead>
<tr>
<th>Config.</th>
<th>aa</th>
<th>bb</th>
<th>cc</th>
<th>all</th>
</tr>
</thead>
<tbody>
<tr>
<td>chosen</td>
<td>0.044</td>
<td>0.041</td>
<td>0.048</td>
<td>0.045</td>
</tr>
<tr>
<td>all data</td>
<td>0.041–0.047</td>
<td>0.040–0.049</td>
<td>0.034–0.054</td>
<td>0.034–0.054</td>
</tr>
</tbody>
</table>
E.2. EXPERIMENTAL RESULTS

E.2.2 Wall

Figures E.3(a)–(c) show static wall pressure distributions for the three configurations. Lines are drawn through the pressure tap values whereas the symbols denote the nearest static pressure readings from the 3-hole probes to the respective wall. Dashed lines are ideal pressure distributions obtained from normal areas to an assumed streamline. Three data sets are shown on each figure: (i) the chosen set whose numerical quantities were selected for performance evaluation and the (ii) minimum- and (iii) maximum-Reynolds number tests for the given configuration. The distributions were virtually identical with any deviations falling within the uncertainty limits. The discrepancy in the two OW pressures for Config. bb nearest to \( x = 0 \) was a consequence of the diffuser flange being slightly smaller than the plexiglas diameter: the left side of the duct was flushed and gave less negative pressure readings on the SW pressure taps. A small lip protruded into the flow on the NE side and provided a locally larger area ratio. The NE data was more characteristic of the CFD results. The absence of appreciable deviations in pressure tap readings at the same axial locations that cannot be attributed to other geometric issues meant that the tap bias was minimal.

In addition to pressure taps placed on the CB, as shown in Fig. 3.4, wool tufts were also located in similar axial locations. During experimentation, tufts were either characterized as attached, fluttering (indicating the onset of separation), or reversed. Centre-body separation points were recorded from wool tufts that occurred at \( x/D_o = 0.57, 0.48, \) and 0.22 for Configs. aa, bb, and cc respectively. These values were in qualitative agreement to the levelling off of the CB pressure distributions in Figs. E.3(a)–(c).

Tufts were placed in approximately 0.1\( D_o \) increments, which was not sufficient resolution to confirm the precise experimental separation point. Furthermore, Wood and Higginbotham [7] observed that neither wall pressure nor tufts are completely reliable indicators of separation. Regardless, the observations were useful in establishing that the flow remained attached to the CB longer for smaller area ratio diffusers without swirl.

With particularly wall pressure distributions for OWa and OWh, good agreement existed between the ideal and outer wall distributions for \( x > 0.1D_o \). Sudden geometry change occurred at point \( S \) due to the kink and incurred a local pressure penalty that was largely responsible for the upstream deviation. Curvature to gradually transition between components would improve the distribution.
Figure E.3: Influence of mass flow rate on configurations with S0 swirl static wall pressure distributions.

E.2.3 Outlet

Figures E.4(a)–(d) show outlet total pressure contours and normal velocity vectors non-dimensionalized by $\langle V_e \rangle$ for the selected configurations. The core of Config. nc was blanked since flow angles exceeded the 7-hole probe calibration range (indicating reversed flow). Symmetric outlet profiles were expected in the axisymmetric diffusers. An investigation of the asymmetry was completed in App. A.5 and concluded that the effect of the asymmetry was tolerable. Similar wall pressure readings from the diametrically opposite taps shown in Figs. E.3(a)–(c) substantiated the quality of the pressure data. Additional verification in App. E.3.6 shows, particularly for Config. aa, agreeable CFD performance predictions and similar outlet velocity profiles to the experimental averaged profiles.

The common cause of asymmetry was attributed to an angular offset between the CB axis and OW axis. This was particularly prevalent with CBa where the core region of minimum velocity was always shifted.
The method of manufacturing the centre-bodies involved circular passes where similar radial asymmetries of \( \pm 1 \text{ mm} \) were measured on all CBs. Possibilities for asymmetry included the CB mounting method, or more likely, the fact that the flow remained attached to CBa longer that resulted in the greater discrepancy. In any event, the contours show that the flow was more uniform for decreasing area ratios since less energy was present in the high velocity region. Vector magnitudes confirmed the absence of any strong secondary flows when a CB was installed.
E.2.4 Reynolds Number Influence

Most exhaust system applications involve fully turbulent flow. To avoid the influence of compressible flow, the apparatus size restricted the Reynolds number range to \( Re_t < 4 \times 10^5 \). Additionally, for any experiment to be credible, the results must be repeatable: biases highlighted in App. A.5 were noticeable during experimentation. These factors were thoroughly scrutinized for Configs. aa, bb, and cc. Tests were typically completed at two or three mass flow rates and then apparatus reassembly occurred to mount a different configuration.

Figures E.5(a)–(c) plot performance against inlet Reynolds number. Two sizes of error bars are particularly evident in Fig. E.5(c) and distinguish between tests where auxiliary experiments were completed (greater uncertainties) to those that omitted the auxiliary experiments. Figures E.5(a) and (b) show that \( C_b \) and \( \gamma \) were constant over the range. Consistency was not obvious in total pressure coefficient; however, Configs. aa and bb did not produce any discernable trends. Table E.3 compares performance from the selected configurations of aa, bb, and cc, to the average and standard deviation in the averages obtained from all tests with the variable speed drive. The tabulated uncertainties include outlet point auxiliary experiments.

<table>
<thead>
<tr>
<th></th>
<th>Config. aa</th>
<th>Config. bb</th>
<th>Config. cc</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>( C_b )</td>
<td>( \gamma )</td>
<td>( C_{p0} )</td>
</tr>
<tr>
<td>chosen test</td>
<td>0.49</td>
<td>0.96</td>
<td>-0.073</td>
</tr>
<tr>
<td>uncertainty, %</td>
<td>0.9</td>
<td>1.8</td>
<td>1.2</td>
</tr>
<tr>
<td>average</td>
<td>0.50</td>
<td>0.95</td>
<td>-0.077</td>
</tr>
<tr>
<td>st.dev, %</td>
<td>1.6</td>
<td>0.7</td>
<td>19.6</td>
</tr>
</tbody>
</table>

Outlet velocity uniformity error bars were not included with Config. cc in Fig. E.5(b) for clarity, since they were on the order of 9%. Table E.3 however, shows that the average \( \gamma \) was similar with a small standard deviation, which gave confidence in the value.

The random nature of the Config. aa and bb \( C_{p0} \) data in Fig. E.5(c) that correlated to respectively higher standard deviations in Tab. E.3 was attributed to the consequence of subtracting two large numbers to yield a small number. On an absolute basis, The deviation in \( C_{p0} \) was less than 0.02 for the two smaller OWs. Confidence in \( C_{p0} \) equivalence was instilled from the Config. cc results and would be better served by reporting absolute uncertainties.
E.2. EXPERIMENTAL RESULTS

Figure E.5: Influence of Reynolds number on experimental performance in configurations with S0 swirl. Hollow symbols denote tests that were completed with valve control.
Although the tested Reynolds number range may be too low to confirm fully turbulent flow, similarities existed in wall pressure distributions and the evaluated performance parameters within the tested range. Literature [9] also supported constant pressure recovery for \( Re_t > 6 \times 10^4 \) so the methodologies implemented to assemble the apparatus, collect the data, and data-set were considered to be good enough provided that the incurred biases were unintentional. A CFD investigation given in App. [E.3.3] simulated flow with higher \( Re_t \) that did not predict Reynolds number independence.

### E.2.5 Mass Flow Comparison

Four axial location traverses were completed during each test (three stations as shown in Fig. 3.4 and the upstream traverse at the annulus inlet). Mass must be conserved in the apparatus since it was characteristic of simple duct flow that did not include secondary entrainment. Table E.4 lists the discrepancies in mass flow rate with respect to the outlet traverse \( \dot{m}_c \). In agreement with the Inlet Traverse Analysis in App. [A.5] the fact that all manual 3-hole traverses were simultaneously completed, did not adversely affect the measured mass flow rate at section ①.

**Table E.4:** Mass flow rate comparison with S0 swirl. Percentage change with respect to \( \dot{m}_c \). Average and maximum rows were based on all configurations with the given outer wall. \( \dot{m}_{Up} \) was calculated from the traverses completed at the annulus inlet.

<table>
<thead>
<tr>
<th>Configuration</th>
<th>aa</th>
<th>bb</th>
<th>cc</th>
</tr>
</thead>
<tbody>
<tr>
<td>( \dot{m}_{Up} )</td>
<td>( \Delta \dot{m}_c )</td>
<td>( \dot{m}_c )</td>
<td>( \dot{m}_c )</td>
</tr>
<tr>
<td>chosen</td>
<td>-0.4</td>
<td>-2.7</td>
<td>-2.6</td>
</tr>
<tr>
<td>OW abs. avg.</td>
<td>2.0</td>
<td>2.6</td>
<td>2.6</td>
</tr>
<tr>
<td>OW abs. max.</td>
<td>3.2</td>
<td>4.6</td>
<td>4.6</td>
</tr>
<tr>
<td>OW std. dev.</td>
<td>0.8</td>
<td>1.3</td>
<td>1.3</td>
</tr>
</tbody>
</table>

Configurations with OWa or OWb had good mass flow agreement, on the most part below 5%. Configurations with OWc produced acceptable agreement where the outlet always calculated a higher rate. In comparison to the smaller diffusers that had smooth contours, Fig. E.4(c) showed a largely fluctuating outlet core region in Config. cc and Fig. E.4(d) showed core flow reversal in Config. nc.

To resolve the problem of separated and rapidly fluctuating flow at the diffuser exit, Wood [119] obtained outlet measurements downstream of a tail pipe that stabilized the flow. A tail pipe was not attached in the
present investigation so questionable velocities were measured in regions with erratic flow. For Config. cc, the 7-hole probe presence may have altered the flow path to give a local forward momentum bias that resulted in higher core flow velocity readings than were actually present—longer sampling times did not obtain more realistic averages. Through omitting the data in the blanked region of Config. nc in Fig. E.4(d) mass flow discrepancy reduced to 2.8%, which agreed with visual observations that flow was reversed in the core region.

### E.2.6 Performance

Figures E.6(a) and (b) plot the objective space for the tested configurations. Error bars depict calculated uncertainties and the solid line represents the Pareto front through Configs. aa, bb, and cc. These trends were in agreement with the predicted performance obtained during the numerical optimization study shown in Figs. B.4(a) and (b): Config. aa had the highest $\gamma$ and best $C_{p_0}$, Config. cc had the largest $C_b$, and Config. bb was the best diffuser assuming equally weighted objectives.

![Graphs showing objective space for configurations](image)

**Figure E.6**: Diffuser geometry study objective space. Like colours denote the centre-bodies whereas like symbols denote the outer walls.

Similarly to Fig. B.4(a), Fig. E.6(a) shows that the Config. na solution obtained higher $\gamma$. Given that three objectives were evaluated, Config. na was excluded from the non-dominated set because its $C_{p_0}$ did not
satisfy the algorithm requirements in that solutions existed with better $C_b$ and $C_{p0}$. Similarly, Config. ba had higher $C_{p0}$ but was excluded due to its lower $\gamma$ than Config. aa.

Pressure recovery coefficient (Eq. (2.2)) trended the same way as back pressure coefficient for the unswirled flow but was on average 4% lower with a CB and 9% lower without a CB.

### E.2.7 Centre-Body Importance

Figure E.7 replots back pressure coefficient from Figs. E.6(a) and (b) with respect to $\alpha_1$ to give a clear indication on the importance of incorporating CBs. For example, instead of a plug-nose, CBN, for OWb, attaching CBb improved $C_b$ by 32.4% and $C_{p0}$ by 47.4% with a minimal 0.7% reduction in $\gamma$. Wood and Higginbotham [7] also reported that adding a CB on a plug nose significantly improved static pressure recovery.

![Figure E.7: Influence of initial expansion angle, $\alpha_1$, on back pressure coefficient for configurations with S0 swirl.](image)

Lines drawn through diffusers that have the same OW in Fig. E.7 showed that CB curvature was also an important design variable. The initial flow angle $\alpha_1 = 14^\circ$ was achieved by Configs. aa and bb and agreed with Johnston [2] as being the optimum annular diffuser expansion angle. The data showed that pressure recovery appreciably declined for $\alpha_1 < 12^\circ$ (comparison of Config. ba to ca) and $\alpha_1 > 19^\circ$ (comparison of Config. cc to bc).

Although the designs of CBA and CBB were subtly different, Config. ba had $\alpha_1 = 12^\circ$ and lost 0.4% in $C_b$ and 0.6% in $\gamma$ but improved 8.2% in $C_{p0}$ over Config. aa. Whereas Config. ab had $\alpha_1 = 16^\circ$ and lost
3.7% in $C_b$, 1.1% in $\gamma$, and 53.1% in $C_{p_0}$ with respect to Config. bb. Note that the confidence of the total pressure loss comparisons was lower owing to the higher uncertainties. As a result, the subtle differences in CB curvature had minimal improvements to performance and thus echoed the findings of Wood and Higginbotham [7]. There were more appreciable changes between CBc and CBb that translated to an 8% loss in $C_b$ from Config. cc to Config. bc.

### E.2.8 Diffuser Effectiveness

Chapter 2 noted that calculating component losses is beneficial for system designs. Figure [E.6(b)] plots total pressure coefficient. Trends with CBs show that losses increased with increasing pressure recovery. For completeness, Tab. [E.5] lists the diffusers effectiveness with comparison to literature. The last two columns list the non-dimensional length and maximum pressure recovery effectiveness at prescribed area ratio, $\eta^{**}$, based on Sovran and Klomp’s [11] diffuser map (Fig. 2.2). At $L/\Delta R = 12$, the maximum pressure recovery coefficient at prescribed non-dimensional length was $C^*_p \approx 0.82$ and corresponded to $AR \approx 3.3$.

**Table E.5:** Diffuser geometry study diffuser effectiveness Eq. (2.7). Non-dimensional length and maximum pressure recovery effectiveness obtained from Fig. 2.2.

<table>
<thead>
<tr>
<th>OW \ CB</th>
<th>n</th>
<th>a</th>
<th>b</th>
<th>c</th>
<th>$L/\Delta R$</th>
<th>$\eta^{**}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>a</td>
<td>0.62</td>
<td>0.80+</td>
<td>0.80</td>
<td>0.75</td>
<td>4.5</td>
<td>0.81</td>
</tr>
<tr>
<td>b</td>
<td>0.60</td>
<td>0.76</td>
<td>0.79</td>
<td>0.77</td>
<td>7.0</td>
<td>0.82</td>
</tr>
<tr>
<td>c</td>
<td>0.49</td>
<td>0.65</td>
<td>0.66</td>
<td>0.72</td>
<td>12.2</td>
<td>0.83</td>
</tr>
</tbody>
</table>

The preferred Configs. aa and bb yielded the highest $\eta$ for the given OW that were in good agreement with the maximum values; however, the manufactured diffusers were longer than the design expectations for the given area ratio or had larger area ratio for the given length. These configurations could be shortened; however, their designs, with negative wall angles, were compared to trends established from annular diffusers with straight or expanding cores with at least $5^\circ$ OW angles and lower inlet blockage.

Configuration cc also had the highest $\eta$ for OWc and essentially ideal nondimensional length; however, its effectiveness was 14% lower than the expected maximum. This meant that the geometry of Config. cc could have been improved to reduce losses.


**E.2.9 Diffuser Sections**

Sectioning the diffuser system as shown in Fig. [5.1](#) into conical expansion and solid fully open outer diameter diffusing components allowed for a better understanding of where the losses occurred. Evaluation of the conical expansion section was based on a plane normal to point $G$. Table [E.6](#) compares static pressure recovery coefficients defined by $C_{p,ce} = (p_G - \langle p_t \rangle) / \langle q_t \rangle$ and $C_{p,sd} = (\langle p_e \rangle - p_T) / \langle q_T \rangle$.

**Table E.6:** Diffuser geometry study static pressure recovery in shaded conical expansion and solid diffusion sections of Fig. [5.1](#). Literature annular diffuser coefficients from Sovran and Klomp [1] and conical diffuser coefficients from McDonald and Fox [138].

<table>
<thead>
<tr>
<th>Config.</th>
<th>Conical Expansion</th>
<th>Solid Diffusion</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>aa</td>
<td>bb</td>
</tr>
<tr>
<td>$\alpha_{ST}$</td>
<td>-7.2°</td>
<td>-5.1°</td>
</tr>
<tr>
<td>$\alpha_{DG}$</td>
<td>-27.0°</td>
<td>-26.1°</td>
</tr>
<tr>
<td>$\hat{L}$</td>
<td>6.2</td>
<td>6.4</td>
</tr>
<tr>
<td>$AR$</td>
<td>1.5</td>
<td>1.7</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>pressure coefficient $C_p$</th>
<th>ideal</th>
<th>literature</th>
<th>experiment</th>
</tr>
</thead>
<tbody>
<tr>
<td>Conical Expansion</td>
<td>0.56</td>
<td>0.65</td>
<td>0.36</td>
</tr>
<tr>
<td>Solid Diffusion</td>
<td>0.41</td>
<td>0.36</td>
<td>0.31</td>
</tr>
<tr>
<td></td>
<td>0.43</td>
<td>0.55</td>
<td>0.74</td>
</tr>
<tr>
<td></td>
<td>0.34</td>
<td>0.34</td>
<td>0.34</td>
</tr>
</tbody>
</table>

The experimental conical expansion downstream pressure component $p_G$ was taken as the average of the pressure taps nearest to the CB TE and the OW pressure taps at the same axial location. The solid diffuser dynamic pressure $\langle q_T \rangle$ was evaluated on a streamline passing through the outlet at $u_e(r) = \langle u_e \rangle$ and from the OW pressure taps immediately before and after point $T$ that defined $q_T$. Non dimensional lengths were $\hat{L}_{ce} = L_{ce} / \Delta R$ where $L_{ce}$ was the average wall length and $\hat{L}_{sd} = x_U / r_T$.

Literature values for the conical expansion section were interpolated from similar annular diffuser geometries with an approximate inlet area blockage $A_B$ of 2% evaluated by Sovran and Klomp [1] and solid conical diffusers by McDonald and Fox [138]. Reasonable comparison existed for Configs. aa and bb; however, the presence of a substantial separated region in Config. cc was why pressure recovery was much worse in the conical expansion section.

Similarly to the analysis of the annular diffuser system in App. [E.2.8](#) diffuser charts indicated that maximum pressure recovery for the conical expansion sections occurred at shorter lengths given the area ratios.
The solid diffusers were well-designed. Higher-than-ideal values for Config. cc were reported because the velocity profile at plane $T$ was likely to be quite distorted.

Analyzing the diffuser sections individually provided information that evaluated the OW constraints defined in App. B.2.1 for the numerical optimization. The conical expansion occupied $1/2$ of the length yet was responsible for 79.8%, 74.4%, and 71.2% of the pressure recovery with respect to $C_p$ for Configs. aa–cc respectively. More diffusion occurred in the conical expansion section; however, its diffuser effectiveness was lower than that calculated for the solid diffusion section. It may have been beneficial to consider configurations with $x_T > D_o$ to determine if better flow uniformity at plane $T$ was beneficial for pressure recovery. There was a tradeoff since the section $x_G < x < x_T$ had decreasing area. The good agreement of $C_{p,sl}$ with respect to ideal suggested that a more aggressive divergence angle, $\alpha_{TU} > 6^\circ$, was possible for wall $TU$.

### E.3 Numerical Results

From the four CBs (a, b, c, n) and three OWs (a, b, c), twelve configurations were simulated using the realizable $k$-$\varepsilon$ turbulence model on coarse ($R k$-$\varepsilon$-c) and fine ($R k$-$\varepsilon$-f) 2D grids and the standard $k$-$\varepsilon$, RNG, and SST models on fine 2D grids without swirl. Similarities existed in the results so the discussion was mainly limited to Configs. aa, bb, and cc (CB|OW). For completeness, some Config. nb results were shown; however, App. E.2.7 emphasized the importance of including CBs in annular diffusers for improved performance. A description of the flow through the diffusers is provided in Sec. 6.1.1.

#### E.3.1 Residual Convergence

Figures [E.8(a)] and [E.8(b)] show the residual history for the two grid types. The characteristic log-linear decay was observed in all equations.

In addition to the equation residuals, all simulations monitored four properties that served as indicators for the performance quantities:

- inlet area-averaged static pressure $\overline{p_i}$
- inlet mass flow-weighted average dynamic pressure $\langle q_i \rangle$
- outlet mass flow-weighted average velocity $\langle u_e \rangle$
• area-averaged axial wall shear stress on the centre-body \( \overline{\tau_{CB}} \)

The first three monitors were properties that were directly relevant for Eqs. (2.1), (2.4), and (2.6). A constant centre-body shear stress could be correlated to the separation point and imply that the significant losses incurred by the recirculation zone had been defined. Figures E.9(a) and (b) show the monitored property history for the two grids. Values were normalized with respect to the quantity obtained at the final iteration with the respective turbulence model: the curves show exponential decay of the errors.

Simulations terminated either when the residual criteria were met or when the maximum number of iterations were reached. Residual targets were not always reached for all configurations because residuals flattened out. Problem cells responsible for the stalled residuals were usually confined to the plenum and not considered to adversely affect the measured performance. For simulations without tabs, sufficient checking of the solution histories ensured that the maximum iteration limits accommodated at least 500 iterations where the monitored quantities had stopped oscillating and were within 0.1% of the final value prior to convergence being declared.
E.3. NUMERICAL RESULTS

Figure E.9: Typical CFD monitored properties history. Values were normalized with respect to the quantity obtained at the final iteration with the given turbulence model.

E.3.2 Coarse Grid Resolution

Five geometrically similar coarse 2D grids were created for Config. bb with S0 swirl. Figures E.10(a)–(b) shows the progression of axial velocity and turbulence kinetic energy profiles and Fig. E.11(a) shows the wall pressure distributions within the Config. bb diffuser on the five grids. Either the average axial spacing and/or average radial spacing was adjusted and resulted in the first cell wall distance distributions shown in Fig. E.11(b). There were no discernable deviations in velocity and pressure; however, the coarser grid predicted lower peaks in the boundary layers—particularly near the CB TE.

Figure E.12 shows the influence of grid density on performance. The coarser grid was rejected due to the discrepancies in \( k \). The finest grid using wall functions was rejected due to its \( y^+ \) values falling below the accepted region for wall function implementation. Although still considered to produce acceptable results, the design of the finest grid was not well suited for the economy of wall function simulation since it was too-refined that resulted in a 15 times greater cell count than the chosen grid. A grid convergence study was completed using the three mid-range grids with cell counts of 26,000, 42,000 and 113,000. The grid convergence index, \( GCI_f \) (Eq. (B.8)), of the chosen grid with respect to the finer grid was below 0.6% in the performance parameters \( C_p \), \( \gamma \), and \( C_{p_0} \).
E.3.3 Reynolds Number Effects

Through specifying a mass flux inlet condition with average mass flux scaling, the five coarse grids evaluated in the previous section were simulated at four additional Reynolds numbers. Additionally, three experimental data sets at different Reynolds numbers were simulated on the chosen coarse grid. Figures E.13(a) and (b) show the resulting simulated and experimental performance. The coarser grid was poorly designed and the finest grid broke down at the lowest Reynolds number. The remaining three grids trended the same up to the $Re_I = 4.5 \times 10^5$ solution: a 10% increase in $C_b$ with respect to the $Re_I = 1.4 \times 10^5$ and effectively constant
values of $\gamma$ and $C_{p_0}$. The amount of flow blockage decreased with increasing Reynolds number.

In App. E.2.4 constant performance was declared over the range of Reynolds numbers largely owing to the overlapping uncertainty error bars. The chosen grid evaluated at $Re_t = 4.5 \times 10^5$ had $GCI_f$ below 2.9% in the performance quantities. The four experimental points of $C_b$, shown in Fig. E.13(a), had an upward slope of $1.7 \times 10^{-7} Re_t$ that was on the same order of magnitude as the chosen grid simulations that increased at $2.3 \times 10^{-7} Re_t$ for $Re_t < 4.5 \times 10^5$. Although there was less certainty that the $Re_t = 6.5 \times 10^5$ solution ($\langle u_t \rangle = 270$ m/s) was grid independent, the change in all performance parameters suggested the rising influence of compressibility effects.
Since industrial applications are typically larger geometries with inlet Reynolds numbers on the order of $10^6$, the CFD predicted more pressure recovery at Reynolds numbers higher than those tested in experiment. If the experiments were not fully turbulent, larger geometrically similar devices would likely have better performance given the same inlet conditions since the development of the boundary layer would result in less proportionate outlet blockage.

### E.3.4 Turbulence Quantity Inputs

Complementary fine 2D grid simulations without swirl were completed on the annulus up to section $\text{t}$. The inlet boundary was supplied with constants for mass-flow, $Tu = 5\%$, and $l = 0.07D_h$ whereas the outlet was defined by an experimentally obtained static pressure $\overline{p}$. First cell wall distance was $y^+ = 1$ with a growth factor of 1.1. Table E.7 lists the calculated pressure performance in the transition duct—based on an area ratio of $A_s/A_t = 1.63$, the ideal static pressure coefficient was 0.62. The choice not to model the airfoil struts was validated given the good agreement in $A_{B,s}$ to experiment. Although CFD was known to predict converging flow fairly well, the $Rk-\epsilon-f$ simulation over-predicted the boundary layer height at section $\text{t}$ whereas the SST simulation under-predicted the boundary layer growth in the transition duct. The larger simulated $A_{B,t}$, however, were responsible for projecting a larger effective area ratio that equated to better-than-expected static pressure loss and higher total pressure coefficients.

Figures E.14(a)–(c) show extracted $Rk-\epsilon-f$ and SST profiles for Config. bb within the diffuser that
were typical for Configs. aa and cc. Comparison was made between the constant $Tu = 5\%$ and the fully-developed turbulence quantities obtained from the complementary simulation to the transition duct inlet. There was clearly a difference in $Tu$ profiles within the transition duct that produced differences in the high-momentum region within the diffuser; however, minimal differences were present in the boundary layers or low-momentum region downstream of the CB.

Figure E.15 shows the influence of the turbulence inputs to the Config. bb wall pressure distributions: the distributions were virtually identical with the only noteworthy variation being in the Rk-$\varepsilon$-f solution that showed the constant $Tu$ input solution estimated slightly lower pressure in the diffuser entrance region. Table E.8 quantifies the influence of the fully-developed turbulent input profiles on performance. With the Rk-$\varepsilon$-f model, the simulation using the fully-developed profile estimated lower performance that could be attributed to thicker boundary layers; however, thicker boundary layers also occurred with SST that did not translate into any appreciable performance differences.

Table E.8: Change in predicted performance due to fully developed turbulent quantity inlet inputs as % of constant $Tu = 5\%$ simulations with S0 swirl.

<table>
<thead>
<tr>
<th>Config</th>
<th>Rk-$\varepsilon$-f</th>
<th>SST</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>$Cb$</td>
<td>$\gamma$</td>
</tr>
<tr>
<td>aa</td>
<td>-2.42</td>
<td>-1.59</td>
</tr>
<tr>
<td>bb</td>
<td>-3.03</td>
<td>-2.62</td>
</tr>
<tr>
<td>cc</td>
<td>-3.98</td>
<td>-4.86</td>
</tr>
</tbody>
</table>

If a $k-\varepsilon$ model was selected, the benefits of a supposedly more accurate pressure distribution with the provided turbulent inlet profiles must be contrasted against the costs associated with obtaining the experimental turbulent data. A 4% discrepancy in $Cb$ was not significant enough in the present investigation to merit acquiring the turbulent data for validation purposes. Alternatively, the SST results demonstrated an insensitivity to the input turbulent profile shapes.
Figure E.14: Input turbulence quantity study predicted Config. bb profiles with S0 swirl.
E.3. NUMERICAL RESULTS

Figure E.15: Input turbulence quantity study predicted influence on Config. bb wall pressure distributions with S0 swirl.

E.3.5 Centre-Body Curvature

Simulations of Configs. aa, bb, and cc were completed on fine grids that applied a $\mathcal{R}/R_o = 0.25$ radius of curvature to point $D$. The axial distance of the filleted corner was $0.04D_o$ (1/4-in). Figures E.16(a)–(c) compare $Rk-\varepsilon$-f and SST profiles for Config. bb of the filleted point $D$ to sharp corner simulations. Virtually no change occurred in the filleted simulations with $Rk-\varepsilon$-f whereas the SST filleted simulation profiles approached those of the $Rk-\varepsilon$-f simulations.

Figure E.17 shows the influence of the filleted corner on axial shear stress distributions. The sharp corner SST simulation predicted an initial separation region downstream of point $D$ that was not predicted by $Rk-\varepsilon$-f or the filleted corner SST simulations. Similar trends were observed for Configs. aa and cc with quantified changes in performance shown in Tab. E.9. For Config. cc, the difference in performance of the SST filleted simulation with respect to the $Rk-\varepsilon$-f sharp corner simulation was half that of the SST sharp corner simulation.

Simulations using the $k-\varepsilon$ model did not require an accurate representation of slight curvatures. Alternatively, the SST model predicted more representative penalties associated with sudden geometry changes. Since the CBs had slight curvatures at points $D$, $E$, and $F$ and the SST model was noted for more accurately simulating boundary layer growth, it may have been beneficial to model the CB curvature.
Figure E.16: Fillet study predicted Config. bb profiles with S0 swirl. A $\Re/R_o = 0.25$ radius of curvature was applied to point D.
E.3. NUMERICAL RESULTS

Figure E.17: Fillet study predicted Config. bb axial shear stress distributions. A $\mathcal{R}/R_o = 0.25$ radius of curvature was applied to point $D$.

Table E.9: Change in predicted performance with S0 swirl due to filleted corner $D$ as % of sharp corner simulations.

<table>
<thead>
<tr>
<th>Config.</th>
<th>Rk-$\epsilon$-f</th>
<th>SST</th>
<th>SST fillet vs. Rk-$\epsilon$-f</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>$C_b$  $\gamma$ $C_{p_0}$</td>
<td>$C_b$  $\gamma$ $C_{p_0}$</td>
<td>$C_b$  $\gamma$ $C_{p_0}$</td>
</tr>
<tr>
<td>aa</td>
<td>0.64  0.01 -4.47</td>
<td>8.52  0.90 -37.24</td>
<td>-2.39 -1.44 -7.41</td>
</tr>
<tr>
<td>bb</td>
<td>0.42  0.12 -3.35</td>
<td>5.24  1.20 -29.99</td>
<td>-2.73 -2.17 -3.56</td>
</tr>
<tr>
<td>cc</td>
<td>0.64  0.72 -4.03</td>
<td>18.14  5.81 -31.19</td>
<td>-16.84 -11.71 101.54</td>
</tr>
</tbody>
</table>

E.3.6 Comparison to Experiment

Inlet

Figures [E.18(a)–(c)] show experimental and numerical pressure profiles obtained at section ① that were typical of all configurations tested without swirl. The numerical total pressure profiles were in excellent agreement in both shape—with the flow peaking nearer to the inner wall—and magnitude. The SST results were slightly better than the $k-\epsilon$ predictions.

Static pressure was not as well-predicted. Although simulations with the SST model gave the proper magnitude for Configs. aa and bb, they did not predict the static pressure increase at the inner cylinder nor the gradual decrease towards the outer wall. For Config. cc, the SST results over-predicted the amount of losses with respect to experiments that resulted in a 23% lower estimation of $\langle q_t \rangle$ whereas the Rk-$\epsilon$-c and Rk-$\epsilon$-f
solutions over-predicted $\langle q_t \rangle$ by 22% and 30% respectively.

The consequence of specifying a transition duct inlet pressure boundary condition meant that the CFD did not obtain the same mass flow rate measured in experiment. Configurations with OWa were within 5% of experiment, 8% for OWb, and 19% for OWc. With a CB, the $k-\varepsilon$ models over-predicted the mass flow rate whereas the SST model and simulations with CBn under-predicted $\dot{m}$.

Configurations aa, bb, cc, and nb were evaluated with uniform mass flow inlet boundary conditions without swirl on coarse grids. The CFD yielded differences below 1.5% in performance (except Config. cc’s $C_{p_0}$ that was 4.8% lower) with respect to the total pressure inlet condition. Mass flux inlet profiles were simulated on Config. bb and were within 0.5% but predicted $\langle p_0 / q_{t,exp} \rangle$ 8.5% lower. The pressure inlet boundary condition predicted a mass flow rate 4.7% higher than actual.

Wall

Wall pressure distributions are provided in Sec. 6.1.2.

Table E.10 compares CB separation points between experiment and CFD. For SST simulations that predicted flow separation and reattachment downstream of point $D$, the values in Tab. E.10 were based on the
E.3. NUMERICAL RESULTS

recirculation region present at the CB TE that was not necessarily caused by the flow lift-off at point D. The number of tufts was not sufficient to confirm the precise experimental separation point; however, the values were closer to those measured in simulations that used the R$k$-$\varepsilon$ model.

Table E.10: Diffuser geometry study onset of centre-body separation and reattachment points (axial coordinate $D_o$). Experimental values were based on the location of the first fluttering tuft from tufts that were characterized as attached, fluttering, or reversed. CFD values were based on CB axial shear stress plots that defined points where $\tau_x = 0$ and/or the $u(x) = 0$ point on centreline velocity profiles.

<table>
<thead>
<tr>
<th>Config.</th>
<th>separation point</th>
<th>reattachment point</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>aa</td>
<td>bb</td>
</tr>
<tr>
<td>experiment</td>
<td>0.57</td>
<td>0.48</td>
</tr>
<tr>
<td>R$k$-$\varepsilon$-c</td>
<td>–</td>
<td>0.60</td>
</tr>
<tr>
<td>R$k$-$\varepsilon$-f</td>
<td>0.51</td>
<td>0.59</td>
</tr>
<tr>
<td>SST</td>
<td>0.1</td>
<td>0.38</td>
</tr>
</tbody>
</table>

Outlet

Figure E.19 shows outlet velocity profiles for all configurations studied. Good agreement between CFD and experiment occurred for configurations including a CB and OWa or OWb. The higher discrepancy at the centreline for Configs. aa and ab was likely due to the asymmetry at the CB TE since minimum velocity was not observed in the outlet centre. In comparison, Tab. E.10 shows that CFD predicted flow reattachment on the centre-body so the CB shaping irregularity influenced the flow.

Outlet profiles for configurations with OWc or CBn in Fig. E.19 show that the experiments achieved greater core flow recovery than that predicted by CFD. In particular, configurations with OWc (particularly Config. nc) had bulk flow following the outer wall curvature as opposed to diffusing towards the centre. Given that the flow can be treated as a confined jet and assuming a spreading rate of $2^\circ$ [184], the core flow from the annular diffuser inlet should have spread down to $\sim 0.58r_U$ at the outlet for configurations with OWc—the experimental results agree.

The CFD for solutions with OWc can be justified due to radial curvature effects that occurred due to the convex curvature around point T. A local reduction in static pressure extended upstream of point T and caused streamlines upwards. Wall pressure taps could not be located directly at point T but the nearest upstream and downstream points did not show the same pressure reduction trends evident in Figs. 6.3(a)-(c).
Figure E.19: Outlet axial velocity profiles with S0 swirl.
Flow separation did not occur in numerical studies at point $T$; therefore, it was assumed by the comparable OW pressure profiles that minimal-to-no separation occurred in experiment.

**Diffuser Sections**

Table E.11 is a reproduction of Tab. E.6 but includes the CFD estimates. The turbulence models predicted the same recovery in the solid diffusion $T_U$ section that were on par with experiment. CFD calculated flow blockages with $Rk$-$\varepsilon$-$f$ at plane $T$ of $A_{B,T} = 0.083, 0.12, \text{ and } 0.32$ for Configs. aa, bb, and cc respectively. In the conical expansion section, the $k$-$\varepsilon$ predictions were closer to those found in literature whereas the SST values were closer to those measured in experiments. Note that a perfect comparison could not be made since the experimental $p_G$ and $p_T$ were based on wall values whereas the CFD calculation used mass-flow averaged plane quantities—the $Rk$-$\varepsilon$-$f$ Config. bb simulation gave $\langle p_T \rangle / p_T \approx 0.83$. In any event, owing to the wider range of turbulence model estimates for $C_{p,ce}$, the flow transport in the conical expansion section was more problematic for CFD to predict.

<table>
<thead>
<tr>
<th>Config.</th>
<th>$C_{p,ce}$</th>
<th>$C_{p,sd}$</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>aa  bb  cc</td>
<td>aa  bb  cc</td>
</tr>
<tr>
<td>ideal</td>
<td>0.56  0.65 0.81</td>
<td>0.41 0.36 0.31</td>
</tr>
<tr>
<td>literature</td>
<td>0.43 0.55 0.74</td>
<td>$\sim 0.35 \sim 0.35 \sim 0.35$</td>
</tr>
<tr>
<td>experiment</td>
<td>0.37 0.41 0.43</td>
<td>0.34 0.34 0.36</td>
</tr>
<tr>
<td>$Rk$-$\varepsilon$-$c$</td>
<td>0.44 0.52 0.59</td>
<td>0.33 0.30 0.31</td>
</tr>
<tr>
<td>$Rk$-$\varepsilon$-$f$</td>
<td>0.45 0.53 0.63</td>
<td>0.32 0.29 0.30</td>
</tr>
<tr>
<td>SST</td>
<td>0.35 0.44 0.34</td>
<td>0.32 0.30 0.29</td>
</tr>
</tbody>
</table>

**Performance**

Figures E.20(a) and (b) plot the objective space for the 12 geometries studied. Only one $k$-$\varepsilon$ solution set was shown because the four $k$-$\varepsilon$ simulation sets predicted similar results. The maximum difference between the coarse grid and fine grid $Rk$-$\varepsilon$ results for the same configuration was 5.1% for $C_b$ and $\gamma$ and 17.6% (OWc) for $C_{p_0}$—most cases were within 4% for all objectives. The predicted performance of the std and RNG $k$-$\varepsilon$ solutions was on average within 4% of $Rk$-$\varepsilon$-$f$. 
Figure E.20: Diffuser geometry study objective space. Like colours denote the CBs whereas like symbols denote the OWs. Small symbols with error bars (95%) are experimental results, medium-sized symbols are results using R$k$-$\epsilon$-$c$, and large symbols are results using the SST turbulence model. Lines drawn for convenience of identifying Configs. aa, bb, and cc (filled symbols) from the various solution sets.

For configurations with a CB, the R$k$-$\epsilon$-$c$ solution predicted $C_b$ 14–20% greater than that measured experimentally. Reasonable agreement was observed in the diffuser wall pressure distributions (that largely discounted not modelling the CB curvature or the nut at the trailing edge), CB separation point, and outlet profiles with OWa or OWb. Up to 3% of the difference may be due to the specification of the constant $Tu$ input at section $\S$ and potentially 6% more if the wind tunnel had $Tu < 5\%$ (recall Fig. 4.2).

The CFD predicted good estimates of performance, particularly for configurations with OWa and OWb. Closer predictions were obtained with the SST model; however, R$k$-$\epsilon$-$c$ was on par, particularly if the larger experimental uncertainties associated with $\gamma$ and $C_{p_0}$ were taken into account. Discrepancies could not be attributed to the summation of experimental minor errors. The evidence suggested that the differences were associated with the calculation of the turbulent quantities where the SST model predicted more turbulent flow in the core region. The higher turbulence developed more system losses that yielded lower $\langle q_t \rangle$. Since $\langle p_{0,t} \rangle \approx \langle p_{0,s} \rangle$, $\langle p_t \rangle$ was increased.

Given the good agreement in wall pressure distributions and predictions of $C_b$ and $C_{p_0}$ within 8%, it was not as apparent why Configs. na and nb poorly predicted the outlet velocity profile and $\gamma$. The turbulence
E.3. NUMERICAL RESULTS

models were benchmarked for backward facing step flows so the discrepancy was unexpected.

The experimental-to-CFD performance comparison was not as close for any of the turbulence models for configurations with OWc, which was particularly evident in the outlet velocity profiles. Some of the discrepancy in $\gamma$ was due to evaluating $\langle u_e \rangle$ from the known inflated values of $\dot{m}_e$. If performance for the Config. cc experiment with S0 swirl was calculated using $\dot{m}_t$, $\gamma = 0.81$ and $C_{p_0} = -0.15$.

Although the equivalent expansion angle for OWc was only $3^\circ$ greater than OWb, similar experimental outlet velocity profiles were observed. In comparison, CFD incorrectly estimated the losses caused by this additional diffusion that gave lower outlet centreline axial velocity. It was possible that the turbulence models constants were tuned to identify OWc as being in the flow regime of appreciable stall—this analysis was not sufficient to justify an investigation that modified the turbulence model constants. The $k-\varepsilon$ solutions, that likely over-predicted the size of the recirculation region, still estimated a larger $C_b$ than experiment whereas the SST model over-estimated the amount of total pressure lost in the diffuser.

Table E.12 evaluates the numerical results of all configurations in absolute and relative terms with respect to experimental results. The average of the calculated quantities shows that the SST model agreed better with experiment. The relative analysis was in recognition that Confg. aa, bb, and cc were selected geometries lying on the Pareto front in the optimization study; however, the interchanged configurations were not likely evaluated in the numerical optimization study.

Based on the twelve simulated configurations, Tab. E.12 identifies the configurations that lie on the Pareto front. For std $k-\varepsilon$ and RNG, the set consisted of Confgs. ba, ab, and cc. Configuration bb was discarded in R$k-\varepsilon-c$ due to the better qualities of Confg. ab: 0.3% higher in $C_b$ and 0.1% higher in $\gamma$; however, Confg. bb was marginally better assuming equally weighted objectives. Simulations with R$k-\varepsilon-f$ predicted Confg. ab as being the best with equally-weighted objectives whereas SST predicted Confg. cb. The analysis confirmed that the numerical optimization study had not converged to the ideal solution set so additional iterations were necessary to better define the set; however, the differences in performance quantities were negligible enough between the configurations that close enough preferential shapes had been defined for the given constraints.

Although the R$k-\varepsilon-f$ framework was used in the numerical optimization study, Tab. E.12 notes that the R$k-\varepsilon-c$ solution was just as accurate in predicting the correct relative frontal placement of the configurations determined by experiment and order assuming equally weighted objectives. The consequence of the SST
Table E.12: Diffuser geometry study comparison of computational to experimental results. Absolute percentages calculated as the average difference of $\dot{m}_e$, $C_b$, $C_p$, $\gamma$, $C_{p_0}$, and $M_t$. Relative comparison based on number matching: (1) the same Pareto front determined by the three objectives and (2) the sorted configurations based on equal objective weighting ($\pm$1 rank).

<table>
<thead>
<tr>
<th>OW \ CB</th>
<th>Rk-$\varepsilon$-c n a b c</th>
<th>Rk-$\varepsilon$-f n a b c</th>
<th>SST n a b c</th>
</tr>
</thead>
<tbody>
<tr>
<td>a</td>
<td>6.2 9.1 8.4 14.0</td>
<td>5.6 7.0 5.4 10.9</td>
<td>5.4 5.8 4.1 10.6</td>
</tr>
<tr>
<td>b</td>
<td>10.3 17.6 12.2 16.8</td>
<td>9.0 17.1 10.4 14.5</td>
<td>7.9 9.6 2.8 12.6</td>
</tr>
<tr>
<td>c</td>
<td>18.4 19.4 21.5 18.8</td>
<td>14.4 22.3 25.6 21.7</td>
<td>17.8 21.3 17.4 17.9</td>
</tr>
<tr>
<td>configurations average</td>
<td></td>
<td></td>
<td>14.4</td>
</tr>
<tr>
<td>Pareto front</td>
<td>aa, ab, cc</td>
<td>aa, ab, cc</td>
<td>ba, ca, cb</td>
</tr>
<tr>
<td>front compare</td>
<td>6</td>
<td>6</td>
<td>5</td>
</tr>
<tr>
<td>sort compare</td>
<td>8</td>
<td>8</td>
<td>4</td>
</tr>
</tbody>
</table>

model predicting better performance for configurations with OWa or OWb but worse performance for configurations with OWc made it less suitable for this range of constraints. Given the desire for correct relative placement, future diffuser optimization studies should use Rk-$\varepsilon$-c. Using coarse grids was a particularly attractive conclusion given that it was also the least computationally expensive.

E.4 Conclusions

1. Static pressure recovery of the conical expansion section for configurations with converging outer walls were on par with similar area ratio configurations in literature.

2. Back pressure coefficient, $C_b$, was a competing objective with respect to outlet velocity uniformity, $\gamma$, and total pressure coefficient, $C_{p_0}$. The manufactured configurations achieved their predicted performance capabilities determined in the optimization study.

3. Coarse 2D grids that used the realizable $k$-$\varepsilon$ turbulence model and wall functions predicted relative trends with respect to geometric variations that were closest to those observed from experiment. Solutions obtained with the SST model had the best absolute comparison.
Appendix F

Swirl Study Supplementary

This study evaluated the influence of swirl in the AR = 1.91 diffusers (OWb). A paper on the experimental results was presented [185].

See Sec. 2.2.4 for the literature review.

F.1 Experimental Results

Table F.1 shows the number of tests that were completed for each configuration in the swirl study. Tests without swirl were included for benchmarking purposes. For the most part, each test was completed at three different mass flow rates, corresponding to $Re_t \approx 1.0$, 1.4, and $2.0 \times 10^5$ at S20 swirl. Tests with the middle mass flow setting were chosen for analysis—the chosen test inlet Reynolds numbers are included in the table.

**Table F.1:** Swirl study number of retained tests. All tests were completed with OWb. A chosen test from each configuration was selected for quantitative analysis.

<table>
<thead>
<tr>
<th>CB</th>
<th>n</th>
<th>a</th>
<th>b</th>
<th>c</th>
<th>chosen $Re_t \times 10^5$</th>
</tr>
</thead>
<tbody>
<tr>
<td>S0</td>
<td>4</td>
<td>7</td>
<td>15</td>
<td>3</td>
<td>1.4</td>
</tr>
<tr>
<td>S10</td>
<td>3</td>
<td>3</td>
<td>3</td>
<td>3</td>
<td>1.5</td>
</tr>
<tr>
<td>S20</td>
<td>3</td>
<td>3</td>
<td>8</td>
<td>4</td>
<td>1.4</td>
</tr>
<tr>
<td>S40</td>
<td>3</td>
<td>3</td>
<td>6</td>
<td>3</td>
<td>1.0</td>
</tr>
<tr>
<td>SCV</td>
<td>4</td>
<td>3</td>
<td>3</td>
<td>3</td>
<td>1.4</td>
</tr>
</tbody>
</table>
F.1.1 Inlet

Figures F.1(a)–(e) show the total and static pressure profiles at the annular diffuser inlet (section 1) for each of the centre-bodies with OWb and the given amount of swirl. A consequence of performing the traverses manually that was not as apparent during the S0 tests was that the probe rotated as it was inserted. Subsequently, the obtained swirl angle profiles were physically unrealistic. Instead, the swirl angles taken from full traverses measured by the 7-hole probe, shown in Figs. 3.5(a) and (b) were used. Fortunately, the raw data did calculate average swirl angles that agreed with the swirler used.

Mass flow rates obtained by the 3-hole probes were in reasonable agreement with $m_e$: within 5% when the
inlet flow was not swirled or swirled with straight-vanes, and 10% with the curved-vane swirler. Although the two 3-hole probes obtained individual mass-flows with upwards of 10% differences, the analysis completed in App. [A.3] concluded that accidental asymmetric flow bias was not enough to discredit the performance quantities. Higher discrepancies with SCV were attributed to the 3-hole probes not being calibrated to flows with strong radial components.

The profile shapes were similar at the respective swirl angles in that total pressure peaked at $r/R_o = 0.87$ for S0 and S10, 0.83 for SCV, was flat for S20, and 0.95 for S40. Static pressure for S0, S10, SCV, and S20 decreased gradually linearly with radius whereas it increased for S40. Area blockage at section $A_{B1}$ ranged from 0.04–0.05 with S0, 0.05–0.06 with S10 and SCV, 0.03–0.04 with S20, and 0.04–0.09 with S40.

### F.1.2 Wall

Figures F.2(a)–(f) show static wall pressure distributions with the indicated amount of inlet swirl. The dashed line depicts the ideal unswirled pressure distribution obtained from normal areas to an assumed streamline for Config. bb but adjusted by the outlet pressure obtained with the given swirler. The distributions were similar at all swirl angles. Distributions using CBa and CBb were very similar, lower values with CBc occurred in the conical expansion section, and lowest values along the outer wall occurred with CBn. At the inlet, the three centre-bodies had similar pressure whereas CBn was higher (less negative) at both inner and outer surfaces.

The distributions with CBs had similar characteristics whereby the distributions using S0, S10, and SCV overlapped. With S20 swirl, pressure deviated as it approached the CB TE. The S40 distributions had a rapid decrease (pressure became more negative) on the CB. With S40 swirl, wall pressure was lost in the conical expansion section of the outer wall, and the solid diffuser had no apparent effect on pressure recovery.

If radial pressure gradients were included in the ideal pressure distribution calculation, streamline curvature was negative up to $x \approx 0.7D_o$ and positive thereafter due to the transition around point $T$. From $\partial p/\partial r = \rho v_\theta^2/r$ where $v_\theta$ is the streamwise (i.e. axial) velocity in this context assuming a 2D planar analysis, static pressure should be higher (less negative) on the CB than the OW up to $x \approx 0.7D_o$. Figures [F.2(a), (b), and (f)] show that higher CB pressure occurred with S0, S10, and SCV swirl. The difference between CB and OW pressures was $\lesssim 0.1\langle q_r \rangle$, so the dominant factor driving pressure recovery was area expansion. Subsequently, even with the presence of geometry curvature in the conical expansion section, the usage of the S10 and SCV swirlers that gave a categorically weak swirl, $S_t = 0.3$, had minimal influence from radial
Figure F.2: Swirl study experimental wall pressure distributions with different CBs.
pressure gradients.

With S20 and S40 swirl, Figs. F.2(c), (d), and (e) show that the OW pressure became more positive than the CB pressure around $x \approx 0.5D_o$ and $x \approx 0.1D_o$ respectively, which seems to contradict the conservation of radial momentum. It was true that radial pressure gradients cannot be neglected where $S_t = 0.7$ for S20 and $S_t = 1.9$ for S40; however, there was another more significant factor that caused the rapid decrease in static pressure approaching the CB TE. The negative pressure gradients with S20 and S40 swirl were not attributed to the geometrical reduction in area between the conical expansion and solid diffusion sections since the smaller swirl angles did not capture this feature nor were they due to flow separation given the identification of the longer attached regions identified in Tab. F.2.

**Table F.2:** Swirl study onset of centre-body separation (axial coordinate $/D_o$) based on location of first fluttering tuft.

<table>
<thead>
<tr>
<th>Config.</th>
<th>S0</th>
<th>S10</th>
<th>S20</th>
<th>S40</th>
<th>SCV</th>
</tr>
</thead>
<tbody>
<tr>
<td>ab</td>
<td>0.57</td>
<td>0.57</td>
<td>0.57</td>
<td>0.57</td>
<td>0.57</td>
</tr>
<tr>
<td>bb</td>
<td>0.48</td>
<td>–</td>
<td>0.65</td>
<td>0.65</td>
<td>0.58</td>
</tr>
<tr>
<td>cb</td>
<td>0.64</td>
<td>–</td>
<td>–</td>
<td>–</td>
<td>0.75</td>
</tr>
</tbody>
</table>

For flow between rotating concentric cylinders, the solution to the angular momentum equation is the sum of a forced vortex and free vortex [56]:

$$v_\theta = C_1 r + \frac{C_2}{r} \quad (F.1)$$

for constants $C_1$ and $C_2$. For the straight vaned swirlers, $C_1$ increased as swirl angle increased and $C_1/C_2 \approx 15,000$, 6,000, and -1,200 for S10, S20, and S40. The SCV swirler had $C_1 \approx 300$ and $C_2 = -1$. The ratio varied with area, where at sections $\S$, $C_1/C_2 \approx 900$, 400, 1,200, and 400, for S10, S20, S40, and SCV respectively. The values were determined from a curve-fit that was applied to a central region assumed to be unaffected by boundary layer growth and were included for discussion purposes only since the radial and axial velocity components cannot be neglected from the angular momentum equation.

Even though solid body rotation was primarily responsible for the tangential velocity profile, the derivative of Eq. (F.1) is:

$$\frac{\partial v_\theta}{\partial r} = C_1 - \frac{C_2}{r^2} \quad (F.2)$$
therefore, the free vortex component was influential in determining the slope. Along the CB where the mean radius decreased, the decoupled function $v_\theta(r)$ was expected to increase for $r < 0.65R_o$ when $c_1/c_2 \approx 400$—around point $E$. CFD profiles with S20 swirl, shown in Fig. 6.5(b) confirmed that a local increase in $\theta$ occurred along the CB surface.

Note that through decoupling, $v_\theta(x) \propto x$, so absent of friction, a mechanism already existed in the diffuser to increase $v_\theta$. For large enough swirl, the increase in tangential velocity required an increase in the streamwise component to satisfy conservation of tangential momentum that exceeded the velocity reduction determined by the area increase, resulting in a net gain of convection. This was the cause for the favourable pressure gradients seen in Figs. F.2(c), (d), and (e) on the CB with S20 and S40 swirl.

### F.1.3 Outlet

The impact of the inlet swirl angle on the outlet axial velocity profiles is shown in Figs. F.3(a)–(e) and outlet tangential velocity profiles in Figs. F.4(a)–(e). The experimental apparatus was unable to obtain symmetric flow as shown by the total pressure contours in Figs. F.5(a)–(e) so the velocity profiles were averaged with the error bars depicting one standard deviation at the given radial location. Larger discrepancies were shown in $v_\theta(r)$ with S0 swirl since $v_\theta \approx 0$, so the non-zero readings were a consequence of asymmetric flow and probe alignment bias.

![Figure F.3: Swirl study outlet axial velocity profiles](image-url)
Figure F.4: Swirl study outlet tangential velocity profiles

Figure F.5: Swirl study Config. bb. outlet total pressure contours and velocity vectors
Comparison of Figs. F.3(a)–(e) show that the flow from the curved vane swirler recovered the greatest amount of centreline \((r = 0)\) axial velocity. Using S20 and S40 swirl achieved flow angles in the core region that exceeded the 7-hole probe calibration range. The profiles had similar outer wall boundary layer growth with maximum axial velocity occurring near \(r \approx 0.6r_e\), except for S40 profiles that peaked at \(r \approx 0.9r_e\). Outlet swirl angles were similar to those measured at section \(\S\) and resulted in swirl numbers of \(S_c = 0, 0.13, 0.33, 0.65,\) and 0.13 for S0, S10, S20, S40, and SCV respectively. The differently shaped profile for S40 was attributed to the larger presence of centrifugal forces and resulting radial pressure gradients.

Similarities existed in Figs. F.3(a)–(e) regardless of the CB used for all swirl angles: CBs a–c had very similar profiles transferring the bulk of the momentum and the maximum velocity of CBn was shifted towards the outer wall. With S40 swirl, the absence of a centre-body had no discernable influence on the axial velocity profile.

### F.1.4 Performance

**Pressure Recovery**

Figure F.6 plots diffuser pressure recovery coefficient against swirl number. Lines are drawn through the straight-vaned swirlers whereas the symbols denote the SCV performance. Pressure recovery coefficient for configurations with a CB at S40 swirl was \(C_p = -0.02\) and \(-0.12\) without. The negative values meant that the diffusers did not recover pressure. The trends show that pressure recovery was essentially constant with S0 and S10 swirl and followed by a rapid loss. The SCV swirler provided the best recovery. These observations were in agreement with those discussed in Sec. 2.2.4 where moderate improvements to pressure recovery occurred with some swirl, but strong swirl resulted in degraded performance. Additionally, there was very little difference between the three CBs that all yielded significantly better recovery at all tested swirl angles than not including one.

Figure F.7 plots back pressure coefficient against the initial expansion angle. The thin solid lines denote OWa and OWc configurations with S0 swirl (or refer to Fig. E.7). Although the diffuser did not generate as much recovery with S20 swirl, its \(C_b\) was on par to the values with less swirl. For S0, S10, S20, and SCV, CBb generated 1–4% higher \(C_b\) showing that the preferential initial flow angle \(\alpha_1 = 14^\circ\) also occurred with swirl. Through, test repetition, there was clearly an improvement in \(C_b\) using CBb over CBc; however, some overlap occurred between the CBb and CBa coefficients and wall pressure distributions, as seen in
F.1. EXPERIMENTAL RESULTS

Figure F.6: Swirl study pressure recovery vs. swirl number

Figure F.7: Swirl study back pressure coefficient vs. initial expansion angle

Table F.3 quantifies the outlet velocity uniformity coefficients for CBb. The calculated γ for the other CBs were within the 2% measurement uncertainty of CBb. The maximum deviation of all tests with the same parameters from the selected configuration was 4.9%. Outlet axial centreline velocities are shown in Fig. F.8. No readings were obtained using the S20 and S40 swirlers that had blanked core regions of 0.18\(r_e\) and 0.37\(r_e\) respectively. From the three CBs at the smaller swirl angles, higher \(u_{cl}\) values were obtained with CBa.

<table>
<thead>
<tr>
<th>swirler</th>
<th>S0</th>
<th>S10</th>
<th>S20</th>
<th>S40</th>
<th>SCV</th>
</tr>
</thead>
<tbody>
<tr>
<td>γ</td>
<td>0.93</td>
<td>0.94</td>
<td>0.91</td>
<td>0.71</td>
<td>0.95</td>
</tr>
</tbody>
</table>

Through evaluating standard deviations of the velocity profiles, profiles obtained during the CBa and CBn tests were less peaky. Small improvements in uniformity likely existed since there was more space available in the diffuser for mixing. S40 swirl was an exception since the CB shape had little influence on the outlet profile, as seen in Fig. F.3(d). The evidence suggests that, from the three CBs, the least amount of outlet distortion occurred in Config. ab.
Total Pressure Loss

Figure F.9 plots total pressure coefficient against the initial flow expansion angle. Similarly to the observations made for static pressure recovery and outlet velocity uniformity, the SCV swirler gave the least amount of total pressure loss and including a CB reduced the amount of total pressure loss. The total pressure coefficients were similar with S0, S10, and S20 swirl. With S40 swirl, $C_{p0}$ was significantly worse.

Figure F.9 shows that tests using CBc or CBb resulted in less total pressure loss with S0, S10, and SCV swirl. Separation points listed in Tab. F.2 add evidence of lower losses with these two CBs since the flow generally remained attached further downstream. Provided that boundary layer separation losses were more significant than those associated with a CTRZ, better $C_{p0}$ was expected with CBc.
F.1.5 Swirl Induced Vorticity

Figures F.10(a)–(e) show outlet axial vorticity (Eq. (2.37)) contours and in-plane streamlines for Config. bb that were characteristic of the other configurations for the given amount of swirl. Without swirl, the flow was largely irrotational with minimal cross flow as noted by the insignificant vectors in Fig. F.5(a). The streamlines were an artifact of the bias in probe alignment. It was uncertain if the peaks in the core region were noise or if the nut holding the CB in place, for example, was responsible for shedding vortices.

Figure F.10: Swirl study outlet axial vorticity contours and in-plane streamlines for Config. bb (looking into flow).

With S10, S20, and S40 swirl, Figs. F.10(b)–(d) show increased strength in CCW motion in the core
region. Flow closer to the OW with S10 and S20 swirl were still largely irrotational with noise attributed to the asymmetric flow profiles, as shown in Figs. F.5(b) and (c).

Through inducing rotational particle motion, the streamlines in Fig. F.10(e) exhibited different characteristics due to SCV, whose blades were oriented CW, in comparison to the S10 solution streamlines shown in Fig. F.10(b). Comparison of tangential velocity profiles, Fig. F.4(b) to F.4(e), shows that the test with SCV had slower moving fluid in the core region. From Tab. F.2, flow separation occurred sooner with SCV, so the apparent vortex pair in the core region may be attributed to the consequence of regaining flow direction with less energized secondary motion.

F.2 Numerical Results

Four configurations defined by the four CBs (a, b, c, n) and OWb were simulated using the realizable $k$-$\varepsilon$ turbulence model on coarse and fine 2D grids and the standard, RNG, and SST models on fine 2D grids with S10, S20, S40, and SCV swirlers. A description of the flow through the diffusers with S20 swirl is provided in Sec. 6.2.1.

F.2.1 Turbulence Models

Figures F.11(a)–(c) show profiles from solutions with the different tested turbulence models in Config. bb with S20 swirl. All turbulence models predicted similar profiles. The shapes of the S20 solution profiles were characteristic of those obtained with S10 and SCV swirl.

Figures F.12(a)–(c) show profiles from solutions with the different tested turbulence models on Config. bb with S40 swirl. The different turbulence models obtained different predictions of the CTRZ due to the stronger amount of swirl: the standard $k$-$\varepsilon$ and SST solutions suppressed the amount of core separation. Given that the R$k$-$\varepsilon$ and RNG models were modified to account for swirl in the turbulent quantities, they were considered to be the more representative solutions.

F.2.2 Fine Grid Resolution

Configuration bb was evaluated at four grid densities using the R$k$-$\varepsilon$ model with enhanced wall treatment and SST turbulence model with S20 swirl. Since the aim of a grid study is to achieve consistent predictions
Figure F.11: Swirl study predicted Config. bb profiles with S20 swirl.
Figure F.12: Swirl study predicted Config. bb profiles with S40 swirl.
of the steepest gradients, a representative geometry in the influence of some swirl was necessary. There were not any dramatic geometric differences in the configurations, so by extension, the findings of this study were applicable to all fine grid simulations with at least S20 swirl and were extrapolated to include the S40 simulations.

Figures F.13(a)–(c) show profiles using the SST turbulence model from the four simulated grids. Minimal deviation from the four simulations was observed in the flow transport through the diffuser. The profiles were representative of those calculated by the other turbulence models. If anything, the coarser grid simulation predicted slightly higher $k$ in the wake of the CB.

Figures F.14(a) and (b) show wall pressure and $y^+$ distributions. All grids converged to the same wall pressure distributions. It was particularly challenging to obtain a sufficient grid resolution near the CB given the locally slower moving fluid; however, $y^+ < 5$ was achieved for the three finest grids. The numbers along the walls in Fig. F.13(a) denote the number of grid cells in the boundary layer at the given plane (turbulent Reynolds number below 250) determined by the chosen grid solution with $R_k$-$\varepsilon$-$f$. Overall, the chosen, finer, and finest grids had satisfactory cell density within the boundary layers.

Figure F.15 plots the performance against grid density. Excellent agreement was observed in $C_b$ and $\gamma$. The apparent deviation in $C_{p_0}$ was a consequence of the figure scaling. Table F.4 quantifies the grid convergence index (Eq. (B.8)) for the three grids with respect to the chosen grid whose average axial spacing was $dx = 0.0022D_o$. The finer grid was included for completeness; however, a grid refinement factor of $r > 1.3$ is typically desired [181], so the values note the shortcomings in comparing two grids with similar cell counts. The GCI for $C_{p_0}$ may be on the high side, but with Figs. F.13(a)–(c) and Fig. F.14(a) showing overlapping profiles, very little value would be added to the study through evaluating grids with additional refinement.

**Table F.4:** Fine grid study Config. bb performance parameter uncertainty with S20 swirl. Uncertainties were calculated with respect to the chosen medium grid.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Cells, $10^6$</th>
<th>change to grid</th>
<th>$r$</th>
<th>$R_k$-$\varepsilon$-$f$</th>
<th>SST</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td>$C_b$</td>
<td>$\gamma$</td>
</tr>
<tr>
<td>chosen grid</td>
<td>0.50</td>
<td>ref</td>
<td>ref</td>
<td>0.63</td>
<td>0.91</td>
</tr>
<tr>
<td>coarser GCI, %</td>
<td>0.23</td>
<td>1.5$dx$</td>
<td>1.50</td>
<td>0.31</td>
<td>0.14</td>
</tr>
<tr>
<td>finer GCI, %</td>
<td>0.63</td>
<td>$y^+/2$</td>
<td>1.13</td>
<td>2.28</td>
<td>0.46</td>
</tr>
<tr>
<td>finest GCI, %</td>
<td>2.00</td>
<td>$dx/2$ &amp; $y^+/2$</td>
<td>2.27</td>
<td>0.07</td>
<td>0.10</td>
</tr>
</tbody>
</table>
Figure F.13: Fine grid study Config. bb S20 SST profiles. The numbers along the walls in Fig. F.13(a) denote the number of grid cells in the boundary layer (with turbulent Reynolds number below 250) at the given plane on the chosen grid.
F.2. NUMERICAL RESULTS

![Graphs showing static pressure and y-plus distributions](image)

**Figure F.14:** Fine grid study Config. bb S20 SST wall distributions.

![Graph showing influence of fine grid density on Config. bb performance](image)

**Figure F.15:** Influence of fine grid density on Config. bb performance with S20 swirl. Solid symbols = Rk-ε-μ solutions, hollow symbols = SST solutions.

### F.2.3 Comparison to Experiment

**Inlet**

Figures [F.16(a)]-[d] show the diffuser inlet axial velocity profiles for Config. bb (CFD non-dimensionalized by their simulated averaged quantities). The CFD accurately captured the magnitude and shape: for example, the slope of the inviscid region with SCV swirl decreased for increasing radius. Figures [F.17(a)]-[d] plot the corresponding swirl angle profiles. Although some differences were observed, it was on the order of the probe alignment bias $B_4$. 
Wall pressure distributions and outlet velocity profiles are provided in Sec. 6.2.2.

Table F.5 lists experimental separation points and the CTRZ extents predicted by the turbulence models for Config. bb at each evaluated swirl angle. Although reattachment points are listed with S40 swirl, those turbulence models predicted a subsequent separated region that extended beyond the plenum outlet. Predicted separation points coincided with the sudden gradient changes near the CB TE in the CB wall pressure.
distributions.

Table F.5: Swirl study Config. bb CTRZ separation and reattachment points (axial coordinate /D_o).

<table>
<thead>
<tr>
<th></th>
<th>separation point</th>
<th>reattachment point</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>S0</td>
<td>S10</td>
</tr>
<tr>
<td>exp</td>
<td>0.48</td>
<td>–</td>
</tr>
<tr>
<td>Rk-ε-c</td>
<td>0.60</td>
<td>0.74</td>
</tr>
<tr>
<td>Rk-ε-f</td>
<td>0.59</td>
<td>0.59</td>
</tr>
<tr>
<td>std</td>
<td>0.59</td>
<td>0.59</td>
</tr>
<tr>
<td>RNG</td>
<td>0.59</td>
<td>0.59</td>
</tr>
<tr>
<td>SST</td>
<td>0.38</td>
<td>0</td>
</tr>
</tbody>
</table>

Performance

Figures F.18, F.19, and F.20 plot the C_b, γ, and C_{p0} performance with respect to swirl angle for Configs. bb and nb. Essentially constant performance was predicted with ≤S20 swirl and the S40 performance was much worse. The CFD coefficients for Configs. ab and cb were similar to those obtained with Config. bb. The CBN simulations always predicted lower performance than simulations including a centre-body.

![Figure F.18](image)

**Figure F.18**: Swirl study Configs. bb and nb back pressure vs. swirl number. Lines drawn through results using flat-vane swirl and large symbols denote the SCV solutions.

![Figure F.19](image)

**Figure F.19**: Swirl study Configs. bb and nb outlet velocity uniformity vs. swirl number.

Comparison between k-ε solutions that used CBa or CBc with up to S20 swirl from its respective CBB solution resulted in maximum deviations in C_b and γ of 1%. Larger differences were observed in the C_{p0}
results where Config. cb always had the lowest penalty for up to S20 swirl. Solutions using the SST model predicted γ within 3% of Config. bb using either CBa or CBc. The Cb with ≤S20 swirl was at least 10% lower in Config. ab and at least 1% higher in Config. cb.

Table F.6 quantifies the cumulative average deviation in performance of solutions with ≤S20 swirl with respect to experiment. The C_{p0} percentages were exacerbated given that the experimental flat-vane swirlers C_{p0} ≈ −0.1. For configurations with a CB, all k-ε model performance predictions with respect to experiment were typical of the values shown; an exception excluded the SCV C_{p0} values that over-predicted experiment by ~ 60%. In agreement with the wall pressure distribution and outlet axial velocity profile observations, the SST model simulations predicted closer C_b and the Rk-ε model predicted closer γ to experiment. For Config. nb, all turbulence models predicted similar performance discrepancies with respect to experiment.

Table F.6: Swirl study Configs. bb and nb variations in predicted performance with respect to experiment, %. Values were calculated as the average of solutions with ≤S20 swirl.

<table>
<thead>
<tr>
<th></th>
<th>Config. bb</th>
<th></th>
<th>Config. nb</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>C_b γ C_{p0}</td>
<td>C_b γ C_{p0}</td>
<td></td>
<td></td>
</tr>
<tr>
<td>k-ε</td>
<td>11 -1.1 -27</td>
<td>-9.2 -11.7 7.6</td>
<td></td>
<td></td>
</tr>
<tr>
<td>SST</td>
<td>3.2 -3.2 28</td>
<td>-9.8 -11.6 -4.8</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Unlike the experimental results, the CFD did not develop any appreciable trends with respect to α_1. Of the 25 simulations (5 swirlers times 5 turbulence models), the highest C_b was predicted by Config. ab 12 times, Config. bb 7 times, and Config. cb 6 times. Configuration ab predicted the highest γ 76% of the time.
and Config. cb the best $C_{p0}$ 80% of the time.

Figure [F.21] compares pressure recovery coefficient against back pressure coefficient. The two coefficients were essentially equal with S0 swirl; however, the larger swirl angles developed negative outlet pressure that exponentially decreased the diffuser’s ability to recover pressure. The solutions with S40 swirl obtained with the RNG and R$k$-$\varepsilon$ models were closer to experiment.

F.3 Conclusions

1. Performance—$C_b$, $\gamma$, and $C_{p0}$—did not significantly depreciate in the $AR = 1.91$ diffusers for swirling inlet flow with swirl number below 0.7 but the more aerodynamic curved-vane swirler with an average 10° angle delivered the best performance and out-performed the straight-vaned 10° swirler.

2. Including a centre-body always provided better pressure recovery. The comparison of three different centre-bodies found that performance based on experiment was independent of swirl angle but each CB had its own strength:

   i. The centre-body that initially provided a flow expansion angle of $\alpha_1 = 14^\circ$ (CBb) gave the most static pressure recovery and 1–4% lower back pressure.

   ii. The flow angle $\alpha_1 = 16^\circ$ using CBa was preferable for higher outlet velocity uniformity.

   iii. The flow angle $\alpha_1 = 12^\circ$ using CBc yielded lower total pressure loss.

   The improvements to $C_b$, $\gamma$, and $C_{p0}$ through minor changes to the centre-body curvature, however, were minimal and thus echoed the findings of Wood and Higginbotham [7].

3. Computational fluid dynamics was successful at obtaining good agreement in wall pressure distributions and outlet velocity profiles with respect to experiment in the $AR = 1.91$ diffusers for $\vartheta_s \leq 20^\circ$. Quantitatively, the SST model predicted closer $C_b$ whereas the R$k$-$\varepsilon$ model predicted closer $\gamma$. With S40 swirl, more representative solutions were predicted using either the R$k$-$\varepsilon$ or RNG turbulence models because of their swirl-dependent modifications.
Appendix G

Configuration bd

This appendix characterizes Config. bd (CB|OW). OWd was not an option in the optimization study and excluded from analysis in the Diffuser Geometry and Swirl studies.

The purpose of the additional outer wall was in response to the Apps. H.2 and H.3 tab studies where the designed diffusers with $AR \leq 2.73$ were determined to be too small. The area ratio for OWd was somewhat arbitrarily chosen given the only requirement that diffuser maps predict substantial stall for its non-dimensional length.

G.1 Experiments with S0 Swirl

Figures G.1(a) and (b) show outlet total pressure contours for Config. bd with no swirl that were based on collecting 5 seconds worth of data for each point at 900 Hz. Two stable flow regimes were observed during operation with flow preference in (a) along the outer wall and (b) the southern quadrant. For type (b), flow preference was towards the bottom as opposed to another quadrant since the OW was tilted with axis pointing slightly downwards. The area ratio for OWd was subsequently in the hysteresis region of diffuser stall maps.

Following the discussion of mass-flow discrepancies in App. E.2.5 the lower pressure regions (core of Fig. G.1(a) and top-half of Fig. G.1(b)) should be blanked. Flow visualization using tufts—Figs. G.2 and G.3 for example—observed reversed flow in the respective regions. Since the data points were not blanked, outlet mass flow rates for the outside- and bottom-flow cases were 67% and 47% greater than $\dot{m}_l$. The 7-hole probe
G.1. EXPERIMENTS WITH S0 SWIRL

data conversion executable did not include an option that described this reversed flow situation where all pressure readings were negative with the highest reading at the centre port. The consequence was that the calculated coefficients described low angle flow.

Figure G.4 shows wall pressure distributions for the two flow regimes. Asymmetric flow for the bottom flow regime case was evident by the deviations in wall pressures on the CB and for $x < 0.5D_o$ on the OW.

It was not apparent if start-up procedure selected a preferential flow regime but the regime could be switched by temporarily blocking the flow (with a hand). Figure G.5 shows single point east manometer measurements taken with the centre port of the 3-hole probe in the middle of the inlet with the SW OW pressure tap nearest the diffuser inlet. As blower speed increased, the flow regime stayed same; however, flow change occurred during the ramp down.

Figure G.6 shows that both flow regimes were possible at all tested flow rates. The shown data were based on the collected manometer measurements. Blower speed was first increased, measurements were taken in the first regime, and then the flow was manipulated to obtain the second regime. Without swirl, the bottom flow regime yielded higher pressure recovery. With swirl, preference was given to only the outside flow regime.
Figure G.2: Config. bd with S0 swirl outside flow regime flow visualization.

Figure G.3: Config. bd with S0 swirl bottom flow regime flow visualization.
G.2 CFD S0 Swirl

Figure G.4: Config. bd with S0 swirl experimental wall pressure distributions. Pressure tap orientations provided for bottom flow test.

Figure G.5: Blower ramp up and down influence on Config. bd.

Figure G.6: Two flow modes in Config. bd.

G.2 CFD S0 Swirl

Figure G.7 plots experimental and CFD wall pressure distributions for Config. bd without swirl. Similarly to Fig. 6.3(c) for Config. cc, the distributions were in agreement in the diffuser but the R$k$-c solution overpredicted the amount of back pressure in the inlet section. Figure G.8 compares outlet axial velocity profiles. The CFD converged towards a solution with the outside flow regime whose profile shape and reversed flow
region extents up to \( r \approx 0.6 r_e \) agreed well with experiment. Magnitudes could not be compared since the experimental \( \bar{\tau}_e \) was inflated due to including the core data that flow visualization confirmed should be blanked.

![Figure G.7: Config. bd wall pressure distributions with S0 swirl.](image)

![Figure G.8: Config. bd axial velocity profiles with S0 swirl.](image)

Table G.1 lists the calculated experimental and CFD performance quantities for Config. bd without swirl. Owing to the questionable \( \dot{m}_e \) values calculated in experiment, outlet mass-flow averaged quantities were evaluated using \( \dot{m}_t \) to obtain the values in the table. Performance based on \( \dot{m}_t \) were \( \gamma = 0.64 \) and \( C_{p_0} = -0.56 \). The peakedness of the velocity profile in Fig. G.8 supports that \( \langle u_e \rangle \) should be higher. Following the trends for diffusers with OWc, it was evident that the Rk-\( \varepsilon \) model grossly over-predicted the wider-angled OWd performance.

**Table G.1:** Config. bd performance with S0 swirl.

<table>
<thead>
<tr>
<th></th>
<th>( C_b )</th>
<th>( \gamma )</th>
<th>( C_{p_0} )</th>
</tr>
</thead>
<tbody>
<tr>
<td>experiment</td>
<td>0.35</td>
<td>0.38</td>
<td>-0.50</td>
</tr>
<tr>
<td>Rk-( \varepsilon )-c</td>
<td>0.52</td>
<td>0.50</td>
<td>-0.37</td>
</tr>
</tbody>
</table>
G.3 CFD with Swirl

Figure G.9 plots experimental and CFD wall pressure distributions for Config. bd with S20 swirl. The distributions were in agreement in the diffuser but the R$k$-$\varepsilon$-c solution over-predicted the amount of back pressure recovery in the inlet section. Figure G.10 compares outlet axial velocity profiles. Similarly to the S0 observations, flow visualization defined a reversed core region of $r \approx 0.6r_e$, so the experimental $\overline{u}_e$ was inflated.

![Figure G.9: Config. bd wall pressure distributions with S20 swirl.](image)

![Figure G.10: Config. bd outlet axial velocity profiles with S20 swirl.](image)

Table G.2 lists the calculated experimental and CFD performance quantities for Config. bd with S20 swirl. Experimental outlet mass-flow averaged quantities were evaluated using $\dot{m}_b$ but equal $\gamma = 0.64$ and $C_{p_0} = -0.56$ if $\dot{m}_e$ was used. In either case, comparison with Tab. G.1 shows that no additional errors were introduced due to the presence of S20 swirl for the wider-angled diffuser.

<table>
<thead>
<tr>
<th></th>
<th>$C_b$</th>
<th>$\gamma$</th>
<th>$C_{p_0}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>experiment</td>
<td>0.35</td>
<td>0.39</td>
<td>-0.51</td>
</tr>
<tr>
<td>R$k$-$\varepsilon$-c</td>
<td>0.52</td>
<td>0.45</td>
<td>-0.36</td>
</tr>
</tbody>
</table>

Table G.2: Config. bd performance with S20 swirl.
Appendix H

Tab Study Supplementary

This appendix includes the analyses of additional tab studies that identified supporting features or similar trends to those presented in Sec. 6.3.

\textbf{H.1} was a full analysis of the CFD and experimental results with no swirl.

\textbf{H.2} was an experimental investigation of tab shapes and placement in Config. bb with S20 swirl and was presented [186].

\textbf{H.3} was an experimental investigation of tabs located at the centre-body base with the four outer walls evaluated with S0 and S20 swirl. The S20 swirl results were presented [187].

\section{H.1 CFD of Tabs with No Swirl}

Four configurations using CBb and OWs a–d were simulated using the realizable $k$-$\epsilon$ turbulence model on a coarse grid that employed wall functions with S0 inlet swirl. Simulations were completed assuming symmetry from four and six equally spaced $15w \times 5h$ tabs located at the centre-body base, resulting in the requirement to model $90^\circ$ and $60^\circ$ wedges.

\subsection{H.1.1 Flow Description}

Figure \textbf{H.1} shows pressure fields and streamlines through the Config. bc diffuser without swirl on a modelled $90^\circ$ wedge for assumed symmetry with $D4 \times 15w \times 5h$ tabs. The labelled cross sections were extracted in...
subsequent figures. Of particular interest was the high pressure bubble just upstream of the tab centered about the $\theta = 45^\circ$ plane and downstream separated region.

![Figure H.1: Config. bc with D4 $\times$ 15w $\times$ 5h tabs and S0 swirl predicted pressure planes and streamlines.](image)

Figures H.2–H.5 plot axial velocity, total pressure, and axial and azimuthal vorticity distributions on axial planes with added normal velocity vectors and streamlines. Just upstream of the tab, at $x = 0$, the main flow was largely irrotational and vorticity in the boundary layers was due to a shearing action. As the flow transported downstream, the tab obstruction pushed the flow outwards, which left a sizeable separated region in its wake. Since static pressure was largely determined by the area ratio that was sub-atmospheric in the diffuser, the slow moving fluid in this separation region caused a significant enough penalty to total pressure to result in negative values, as seen on the $x = 0.13D_o$ and $0.26D_o$ planes in Fig. H.3.

Counter-rotating vortex pairs, seen in the Fig. H.4 distributions, were due to the axial dominated flow path around the tab. The exact progression of the CVP could not be determined from the CFD due to the graphics showing a snapshot of a likely transient phenomenon. The shown structures for Config. bc were
Figure H.2: Config. bc with D4×15w×5h tabs and S0 swirl axial velocity contours and velocity vectors.

Figure H.3: Config. bc with D4×15w×5h tabs and S0 swirl predicted total pressure contours.
Figure H.4: Config. bc with D4×15w×5h tabs and S0 swirl axial vorticity contours and streamlines.

Figure H.5: Config. bc with D4×15w×5h tabs and S0 swirl predicted azimuthal vorticity contours.
representative of the smaller diffusers; however, the tab wake with OWd was more coherent, and the CVP had stable foci and were spreading outwards.

Hairpin structures, shown in Fig. H.5, were predicted to ride on top of the CVP and predominantly spread outwards as they moved downstream. Once the hairpin reached the outer wall, it broke down the shear force present in the boundary layer. The tops were regions of high tangential vorticity and legs were dominated by high radial vorticity. A small reversed hairpin was observed just downstream of the tab. The original strength of the azimuthal structures was at least five times larger than the streamwise structures. Vortex smearing occurred as a consequence of obtaining steady-state solutions to the time-averaged numerical equations.

Turbulence intensity contours in Fig. H.6 showed that the region of high turbulence kinetic energy was of similar structure to the region of high $\xi_a$. Although $\xi_a$ strength exponentially decayed, $Tu$ peaked around the $x = 0.39D_o$ plane. In addition to the induced turbulence caused by the vortex generators, shearing occurred between the high velocity fluid not immediately affected by the tab and the slower moving fluid in the recirculation region to account for the increased $Tu$ levels while the vortex structures were still coherent.

A boundary layer on a flat plate exhibits similar characteristics: a high shear region is induced above the
H.1. CFD OF TABS WITH NO SWIRL

Hairpin vortices, it intensifies, elongates, and rolls up. The high shear layer encounters a cascading breakdown that transitions vortices into smaller units with frequency spectra of measurable flow parameters approaching randomness [188].

Coherent CVP and hairpin structures based on still having 50% of the original strength remained until $x \approx 0.27D_o \equiv 7h$ and $x \approx 0.16D_o \equiv 4h$ respectively for the three smaller diffusers that were in agreement with lengths measured by Habchi et al. [104]. For OWd, survival lengths increased to $x \approx 0.36D_o$ and $x \approx 0.26D_o$.

The vorticity dominated features predicted in this CFD analysis were in agreement with the structures defined in Fig. 2.7 and the investigations of fluid interaction due to lobed mixers in [98-100]. Lei et al. [100] observed from CFD that normal vortices broke down at the streamwise vortex cores and it was the breakdown that was primarily responsible for accelerated jet mixing.

H.1.2 Residual Convergence

CFD with tabs was more problematic. All residuals flattened out so presented solutions were based on the final iteration. Figures [H.7(a) and (b)] show the residual and monitored property histories. The pressure and velocity monitors essentially levelled off to give some confidence that the results converged, but it was particularly evident in the shear stress monitor that sinusoidal behaviour was present. For S0 simulations, upwards of a 5% oscillation in $\tau_{CB}$ occurred with periodic boundary conditions. With symmetry boundary conditions, the range increased to 20%. The simulation of Config. bd with tabs was an exception whose residuals flattened out to below $10^{-6}$ and no oscillations were observed in the monitored properties.

The geometric symmetry about the tab of the mesh was confirmed; however, since simulations were solved with parallel processing, the mesh partitions created with the default Metis method were not symmetric. For the configuration used to create the history figures, a partition occurred at roughly $0.52D_o$ that segmented panel $EF$. The partition surface appeared randomly segmented that was created from three axial rows of cells. Tests were repeated on one computer that took about 160% times longer per 100 iterations but yielded similar monitored-property oscillations and performance within 0.5%.

Reducing the under-relaxation factors lengthened the period and increased the magnitude of the oscillation. Although symmetry boundary conditions forced a gradient to achieve the known symmetric flow, solutions with periodic boundary conditions were selected since periodicity ensured global conservation of mass, thus making the solutions more internally consistent.
Second order accurate discretization schemes were prone to fluctuations that may be exacerbated when cell faces were not aligned to the flow direction since they were not bounded. The QUICK scheme was prone for giving physically unrealistic values and instability for the turbulent quantities when wiggles occur [188]. Grid refinement may have relieved some of the stress in the high gradient regions. Simulations with different solver algorithms may have been worthwhile, particularly since the tabs likely shed alternating vortices.

H.1.3 Tab Wake

Figure H.8 shows the Config. bc extents of the recirculation region in the tab wake on the quadric surface (Eq. (6.1)) that appeared asymmetric. In comparison, Fig. H.9 shows that Config. bd predicted stable symmetric structures. The streamlines for Config. bc show that the shape of the recirculation region was elliptical with major axis roughly 1.8 times longer than the minor axis, giving an eccentricity of $e \approx 0.83$. Shapes predicted by the other OW simulations with four tabs and S0 swirl were similar with eccentricity $e \approx 0.85$ whereas its major-axis length increased by $2a/D_o = 0.094AR + 0.20$.

Figure H.10 is a plot of axial vorticity against axial distance from the created quadric surface data. The magnitude peaks coincide with the circular streamlines in Fig. H.8. The dashed line, $C_{bl} \approx 0$, was linearly
Figure H.8: Config. bc with D4×15w×5h tabs and S0 swirl predicted quadric surface pressure and streamlines.

curve-fit through the data and confirmed that the averages of the CW and CCW circulations were equal. Axial vorticity was effectively damped out on the quadric plane at $x = 0.86D_o$. 

Figure H.9: Config. bd with D4×15w×5h tabs and S0 swirl predicted quadric surface pressure and streamlines.
H.1.4 Comparison to Experiment

Since periodic flow was likely, the implementation of steady-state CFD was questionable; however, the results compared reasonably well to experiment. Having said that, Config. bc results were particularly sensitive to the solution procedure. The implemented procedure of converging a first-order accurate scheme followed by a second-order scheme resulted in a $C_b$ 50% greater than only using the second-order schemes after defining the tab as a wall boundary condition. The solution absent of the first-order discretization iterations was similar to Fig. H.9 that predicted stable CVP. The other configurations were insensitive to iteration procedure that predicted a 1% differentiation in performance.

Inlet

Figures H.11(a) and (b) show Configs. bc and bd inlet pressure profiles with $D4 \times 15w \times 5h$ tabs and S0 swirl. The experimental traverses were taken on the $\theta = 0^\circ$ axis (circumferentially offset from the tabs placed at $\theta = 45^\circ$) and two linear extractions were taken from the CFD data at $\theta = 0^\circ$ and $\theta = 45^\circ$. Total pressure was well-predicted and the CFD profiles showed that the influence of the downstream tab did not affect the

---

**Figure H.10:** Config. bc with 4 tabs and S0 swirl predicted quadric surface axial vorticity. Positive CCW.
axisymmetry of the streamlines passing through section $\theta$.

![Graph](image)

**Figure H.11:** Configs. bc and bd inlet pressure profiles with D4 $\times$ 15w $\times$ 5h tabs and S0 swirl. CFD non-dimensionalized by the experimental $\langle q_t, \text{exp} \rangle$.

Back pressure was under-predicted (more positive) for Config. bc but over-predicted for Config. bd and resulted in 5.9% greater and 8.5% lower mass flow rates respectively. Since the 3-hole traverses were completed from W and E (i.e. $\theta = 0^\circ$), the calculated experimental average static pressure was lower than the actual average. In comparison, the Config. bc extracted CFD averaged pressure on the $\theta = 45^\circ$ plane was 12% less negative than the $\theta = 0^\circ$ plane average. The upstream propagation of static pressure was distorted by the tab presence and meant that the obtained experimental $p_t$ implied better $C_h$ than what actually occurred. The difference in tangential pressure distribution to the experimental performance, however, was insignificant in comparison to CFD incorrectly predicting the dynamic pressure.

**Wall**

Figures [H.12(a)] and [H.12(b)] plot wall pressure distributions for D4 $\times$ 15w $\times$ 5h tabs in Configs. bc and bd. Centre-body taps were placed on the $\theta = 0^\circ$ plane whereas OW taps were inline with the tabs at $\theta = 45^\circ$. Tangential gradients were evident at point D whereby a pressure hill was predicted along $\theta = 45^\circ$ but pressure drop along $\theta = 0^\circ$. The pressure drop was caused by local radial vorticity induced by the tab. Streamlines went
sideways to preferentially avoid the pressure hill and resulted in higher velocity along the inner wall and further away from the tab.

Figure H.12: Configs. bc and bd wall pressure distributions with D4×15w×5h tabs and S0 swirl. CFD non-dimensionalized by the experimental $\langle q_t, \text{exp} \rangle$

CFD of Config. bc provided reasonable magnitude estimates for the corresponding slices in the diffuser that also captured the OW dip at $x \approx 0.2D_o$. The curvature discrepancy around $x = 0.5D_o$ was evident in the flow development from Fig. H.4(d) to (e) that showed a switch from positive to negative axial vorticity. Upstream differentiation between experiment and CFD was attributed to CFD predicting a lower pressure penalty associated with the tab obstruction.

The Config. bd CFD distributions agreed well with experiment. The variation between the NE and SW OW pressure data may be due to the SW tab not being placed exactly inline with the SW taps; this was a reasonable possibility given that the SW distribution was similar to the $\theta = 0^\circ$ prediction.

Outlet

Figure H.13 shows a comparison of outlet $p_0$ and $\xi_a$ between CFD and experiment with S0 swirl for Config. bc with D4×15w×5h tabs. The prediction in the NW quadrant correctly displaced total pressure and captured similar high magnitude strength near the outer wall. The lower-magnitude inner core was not observed in experiment. The experimental axial vorticity plot was effectively just noise that negated the value in determining the experimental $\xi_a$. The CFD contours showed some shear strength remained in the outer wall.
boundary layer but the CVP had dissipated. Conversion to the CFD contours shown in Fig. [H.4(b)] requires
C_{ξ_x}/C_{ξ_x,e} = \frac{D_{h,i}/(u_i)}{r_e/(\langle V_e \rangle)} \approx 0.18.

Figure H.13: Config. bc with D4×15w×5h tabs and S0 swirl outlet contours. CFD results plotted on top
two quadrants and experiment on bottom two quadrants. Total pressure contours on left quadrants and
axial vorticity contours on right quadrants.

Figure H.14 is primarily shown to highlight the predicted features for Config. bd with four tabs. The
discussion associated with Fig. [H.21(b)] concluded that the experimental outlet data were questionable. Ex-
perimental mass-flow averaged quantities were based on \dot{m}_t; however, the magnitudes were still substantially
different from CFD. Blanked experimental regions were defined by the top 20% in normal vector magnitude
whereas CFD predicted reversed flow inline with the tab along the outer wall. The blanked experimental
regions were confirmed with flow visualization that flow reversal occurred at the outer wall downstream of
the tabs and significant fluttering occurred in the core flow region. Between tabs (on the axes), the flow was
attached near the outer wall and coincided with the larger total pressure contours carrying the majority of
the outflow. Since the experimental data did qualitatively capture the predicted flow behaviour due to tabs, it
confirmed that there were not equipment problems but rather that the traverse location was unsuitable.

The plotted CFD contours in Fig. H.14 show that a significant amount of reversed flow was entrained at
the outlet. In comparison to Fig. H.13 that showed the CVP had largely dissipated at the Config. bc outlet,
the Config. bd ξ_x contours showed that strength remained in the CVP and that a smaller second set formed
by the reversed flow was equally strong. By re-examining Fig. [H.9] the flow description for Config. bd with four tabs can be completed: the reversed flow entering through the outlet passed through favourable pressure gradients and energized the separated region in the tab wake. This additional energy helped to stabilize the recirculation region.

**Performance**

Figure [H.15(a)] plots the back pressure performance versus area ratio for the unswirled diffusers with none, 4, and 6 equally spaced 15w×5h tabs at the CBb base. The shown 0-tab CFD solutions were from 90° wedge simulations that, like in App. [E.3.6], over-predicted \( C_b \) with \( Rk-\varepsilon \) by 15–20% in OWs a–c and 50% in OWd. With tabs, CFD predicted similar trends with respect to AR whereas experiment obtained maximum performance for Config. bd with 4 tabs. Given the reasonable agreement in wall pressure, it would be worth obtaining better quality outlet data to determine if the predicted amount of entrained flow at the outlet was realistic. The fact that the experimental 0-tab \( C_b \) overlaps with the CFD 4-tab \( C_b \) was entirely coincidental.

Figures [H.15(b)] and [c] plot the \( \gamma \) and \( C_{p0} \) performance with S0 and tabs. The shown experimental values were calculated assuming \( \dot{m}_t \) in the mass-flow averaged quantities and larger error bars denote tests that included auxiliary experiments. For OWs a–c, CFD agreed with experiment that \( \gamma \) increased and \( C_{p0} \)
decreased with additional tabs. Improvements were not observed in the Config. bd CFD performance with tabs. It was curious that $C_{p_0}$ agreed for the completed tests with tabs; however, there was too much uncertainty associated with the quality of the experimental outlet data to substantiate this agreement. Outlet traverses for tab experiments with swirl were better quality.

### H.2 Config. bb Experiments with Tabs

Table H.1 shows the number of tests that were completed for each tab configuration placed in Config. bb with S20 swirl. Tests with no tabs were included for benchmarking purposes. In general, each configuration was tested at two different mass flow rates. Although Config. bb was a relatively well-designed diffuser, there was a CTRZ present in the diffuser from the strong S20 swirling inlet flow. This flow phenomena was comparable to flow separation that occurs in wide-angled diffusers so it was hypothesized that augmentation devices
would be beneficial for recovering the energy lost in a CTRZ. The benefits of the well-designed diffusers provided for a decoupled analysis to characterize the tab vorticity with respect to the swirling flow.

Table H.1: Config. bb tab study number of retained tests.

<table>
<thead>
<tr>
<th># tabs</th>
<th>$5w \times 5h$</th>
<th>$5w \times 8h$</th>
<th>$15w \times 5h$</th>
<th>$15w \times 8h$</th>
<th>$25w \times 5h$</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td></td>
<td></td>
<td>$-8$</td>
<td>$-8$</td>
<td></td>
</tr>
<tr>
<td>1</td>
<td></td>
<td></td>
<td>1</td>
<td></td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>2</td>
<td></td>
<td>2</td>
<td>2</td>
<td>2</td>
</tr>
<tr>
<td>3</td>
<td></td>
<td></td>
<td>2</td>
<td></td>
<td></td>
</tr>
<tr>
<td>4</td>
<td>2</td>
<td>2</td>
<td>11$^a$</td>
<td>10$^b$</td>
<td>2</td>
</tr>
<tr>
<td>6</td>
<td></td>
<td></td>
<td>2</td>
<td></td>
<td></td>
</tr>
<tr>
<td>8</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>2</td>
</tr>
</tbody>
</table>

$^a$ included tests with 90° orientation (2), reversed tabs (2), and on outer wall (2)
$^b$ included tests on CB at $x = 0$ (3), tuft1 (2), $x_E$ (3), and $x_F$ (2).

H.2.1 Tab Orientation Selection

Figure H.16 plots wall pressure distributions using (location-number×width×height) D4×15w×5h tabs at two orientations on the CB base (point D): the red data (‘X’ pattern) placed the tabs at the SE, NE, NW, and SW quadrants looking upstream (see Fig. 6.21(e)) whereas the green data (‘+’ pattern) located the tabs at the S, E (right), N (top), and W quadrants. Error bars are calculated uncertainties and the long-dashed line is the ideal pressure distribution calculated from normal areas to an assumed streamline in unswirled flow. The selected instrumentation orientation was shown in Fig. 3.7: pressure taps along the centre-body and the three-hole traverses were oriented E and W whereas the outer wall pressure taps were oriented SW and NE (with several duplications SE and NW).

The CB ‘+’ and OW ‘X’ orientations had lower pressure than the corresponding readings that were circumferentially offset 45° from the tabs; therefore the tabs caused a low pressure region immediately downstream that extended to $x \approx 0.35D_o$. Beyond $x \approx 0.35D_o$, the OW pressures from the two tab orientations were similar; however, the CB pressure readings from the taps downstream of the tabs were slightly lower.

The presence of the tabs propagated upstream and biased the static pressure readings obtained by the 3-hole probes. Figure H.17 shows static pressure profiles at the annular diffuser inlet section obtained from the two orientations: the probes in-line with the tabs (‘+’ orientation) were slightly higher than the ‘X’
Figure H.16: Tab orientation influence on wall pressure distributions with $D4 \times 15w \times 5h$ tabs and S20 swirl. CB taps inline with ‘+’ pattern and OW taps inline with ‘X’ pattern.

The upstream and downstream features were evidence of the tab induced pressure hill. Zaman, Reeder, and Samimy [109] remarked that the hill was the main contributor to vorticity generation that was responsible for energizing the boundary layer.

Table H.2 confirms that higher objective values with better mass-flow agreement occurred during the ‘X’ pattern traverse; therefore, tabs were not placed immediately downstream of the 3-hole probes for the remaining configurations tested.

Table H.2: Impact of tab orientation on measured inlet quantities.

<table>
<thead>
<tr>
<th>Config.</th>
<th>‘X’ pattern</th>
<th>‘+’ pattern</th>
</tr>
</thead>
<tbody>
<tr>
<td>$C_b$</td>
<td>0.46 ± 0.01</td>
<td>0.43 ± 0.01</td>
</tr>
<tr>
<td>$\gamma$</td>
<td>0.95 ± 0.01</td>
<td>0.94 ± 0.01</td>
</tr>
<tr>
<td>$C_p0$</td>
<td>-0.24 ± 0.01</td>
<td>-0.26 ± 0.01</td>
</tr>
<tr>
<td>$m_t$-$m_e$</td>
<td>-2.0%</td>
<td>-5.0%</td>
</tr>
</tbody>
</table>

Figure H.17 also shows that the downstream influence of the tabs did not appreciably impact the total pressure profiles in that boundary layer area blockage was $A_{Bf} = 0.026–0.039$ for all Config. bb tests with tabs. The inner wall boundary layer was approximately 5 mm, $\delta_i \approx 0.06R_o$, and confirmed that the smaller tab height, 5h, was appropriately sized based on the advice given in Sec. 2.4.1 to induce mixing.
H.2.2 Tab Axial Location Study

Tabs were tested experimentally on the CB at axial locations $x = 0.13D_o$, $x_E$, and $x_F$. To compensate for the larger cross section area downstream, a numerical investigation determined that the CB boundary layer thickness for $x < x_E$ was $\delta < 9$ mm (see Fig. 6.5(a) for example) so $4 \times 15w \times 8h$ tabs were used. At the inlet, $A_{tab} = 0.07$ whereas the tab area blockage reduced to $A_{tab} = 0.04$ at $x = x_F$.

Figure H.18(b) shows for $A_{tab} > 0.04$, that tabs placed at the diffuser inlet had positive $u_{cl}$ whereas no centreline velocities were recorded with tabs placed on the CB. A CTRZ extended beyond the outlet plane when tabs were placed on the CB and the equivalent radius of blanked data was $0.13–0.16r_e$. It was apparent that tabs placed on the CB were not as beneficial for mixing due to the poorer performance in $\gamma$ and $C_q$. The reason that the tabs placed on the CB were not beneficial was clear from Fig. H.18(a) because $C_{p_0} \approx -0.1$ was similar to the no-tab configuration: the boundary layer thickness at even $x = 0.13D_o$ must exceed $8h$ so the tabs did not interact with the flow. For tabs placed on the CB to get the same impact as tabs placed at the CB base, a larger tab height was necessary.
H.2. CONFIG. BB EXPERIMENTS WITH TABS

Figure H.18: Config. bb tab performance with S20 swirl. Projected tab blockage area ratios $A_{tab}$ were non-dimensionalized by the normal area at the given axial location to an assumed streamline. The solid lines pass through the results from varying the number of tabs at the CB base (point D). Linear curves were fit to the pressure coefficients with the goodness of fit evaluated by the $r$-square value. The curve-fit to the dynamic pressure was based on the 0–2 tab data only. Shown uncertainty error bars were typical for the other cases. The long-dashed lines identify the same tabs that were implemented at the various axial locations on the CB (axis included in Fig. H.18(a)). The symbols show the remaining configurations tested.

H.2.3 Tab Width and Height Study

The 5w and 25w tabs were tested at point D. Figure H.18(a) shows that the two pressure coefficients follow the same trendlines with respect to tab blockage area fraction defined by the 15w results. For the attempted 5w tab configurations, $A_{tab}$ never exceeded the tab blockage of the D2 × 15w × 5h case. The results were $u_{cl} = 0$ and no improvements to $\gamma$ relative to the no-tab test.

The two 25w tests completed had positive centreline velocity values. The $u_{cl}$ for the D4 × 25w × 5h configuration was 56% greater than the trend developed by the 15w tabs for the same $A_{tab}$ fraction. This resulted in a slightly higher $\gamma$ with a smaller $C_b$ penalty even though the total pressure lost followed the linear trend.
The two remaining data sets in Figs. H.18(a)–(b) placed $4 \times 15w \times 5h$ tabs at the diffuser inlet on the CB base inclined upstream at $\phi = 45^\circ$ (right triangles) and on the outer wall (left triangles). These two configurations had the same projected area blockage as the D4 $\times 15w \times 5h$ configuration; however, centreline velocities were not recorded and there was lower $\gamma$. That these orientations provide less mixing capability to improve $C_q$ was evident from the lower pressure penalties.

**H.2.4 Tab Induced Vorticity**

Figures H.19(a)–(c) show axial vorticity contours and streamlines from three configurations. The core regions of the plots were dominated by high CCW vorticity due to the more significant slope differences in $v_\theta(r)$ with respect to $u(r)$, as shown in Figs. 6.22(a) and (b). The in-plane streamlines were concentrated at $r \approx 0.45r_e$. This radius loosely coincided with the radius where maximum tangential velocity occurred in Fig. 6.22(b). For $r < 0.45r_e$, the streamlines circulated inwards. And for $r > 0.45r_e$, the streamlines circulated outwards, which demonstrated that free vortex rotation, likely caused by OW boundary layer growth, was also present.

The apparent regions of strong CCW and CW vorticity for $r > 0.45r_e$, as shown in Fig. H.19(a) was likely noise due to the asymmetric contours, as shown in Fig. 6.21(a). For the 2- and 4-tabs plots in Figs. H.19(b) and (c) there were identifiable high vorticity, $C_{\xi,e} > 0.5$, regions and potential low vorticity, $C_{\xi,e} \approx 0$.
regions. Recalling the counter rotating vortex pair in Fig. [2.7] and looking upstream, CCW motion originated from the left side of the tab and CW motion from the right side. Tab orientation in the diffuser with CCW swirl had the left edge as the trailing edge. Indentations rotated approximately 90° from the initiating tab. At the outlet, the circled positive $\xi_x$ regions were the lagging remnants of the CCW motion and the $\xi_x \approx 0$ regions were the remnants of the CW motion. This observations were confirmed by the CFD. In the presence of S20 swirl, the CVP CW motion generated by the tab could not be maintained.

H.3 All Outer Walls Experiments with Tabs

The Config. bb study with tabs confirmed that the CTRZ was reduced with tabs; however, the amount of kinetic energy recovered with OWb was below the amount of losses caused by placing obstructions in the flow. This study investigated tabs in the larger AR diffusers with S0 and S20 swirl to determine if tabs could improve pressure recovery. Table H.3 lists the number of tests completed that placed tabs at the CB base.

<table>
<thead>
<tr>
<th># tabs</th>
<th>S0 swirl</th>
<th>S20 swirl</th>
</tr>
</thead>
<tbody>
<tr>
<td>Config. ba</td>
<td>3</td>
<td>1</td>
</tr>
<tr>
<td>Config. bb</td>
<td>3</td>
<td>2</td>
</tr>
<tr>
<td>Config. bc</td>
<td>1</td>
<td>1</td>
</tr>
<tr>
<td>Config. bd</td>
<td>7</td>
<td>4</td>
</tr>
</tbody>
</table>

H.3.1 S0 Swirl

Inlet profiles were similar to those depicted in Fig. [E.2] with or without tabs where inlet flow area blockage with S0 swirl was $A_{B,t} = 0.039–0.054$. The number of retained tests listed in Tab. H.3 for Config. bd with S0 swirl included both flow regimes, as discussed in App. [G.3] that even occurred when tabs were present.
Wall

Figures H.20(a)–(c) show static wall pressure distributions for Configs. bc and bd with S0 swirl and the given number of tabs. Dashed lines are ideal pressure distributions obtained from normal areas to an assumed streamline. Centre-body pressure taps were 45° circumferentially out of phase of the tabs. When 4 tabs were inserted, OW pressure taps were directly downstream and reported a pressure drop from $x = 0$ to $x = 0.2D_o$, as seen in Fig H.20(a). The previous section and the 0- and 2-tab distributions showed that the first two taps, when offset 45° from the tabs, gave similar values.

The CB wool tufts were either characterized as attached, forward or reversed fluttering, or reversed during experiment. Table H.4 identifies values that can be compared against features in the pressure distributions. Flow did not stay attached to the Config. bc centre-body. The fluttering regions with 0- and 2-tabs were roughly the same as the lengths of constant pressure on the CB in Fig H.20(a). The region of reversed flow...
observed at the CB TE had increased pressure. All tufts were described as fluttering for the 4-tabs test, so the vortices generated by the tabs must have been effective at energizing the low momentum core. The positive-sloped CB pressure distribution was evidence that flow remained attached. The OW distributions were similar for each case where quantitative differences between the number of tabs implemented were caused by losses incurred by the tabs.

**Table H.4:** All OWs tab study with S0 swirl tuft data (axial coordinate / \(D_o\)). Locations identify Config. bc = reversed flow, Config. bd (outside flow) = rev. flutter.

<table>
<thead>
<tr>
<th>Config.</th>
<th>bc</th>
<th>bd (bottom flow)</th>
<th>bd (outside flow)</th>
</tr>
</thead>
<tbody>
<tr>
<td>0 tabs</td>
<td>0.48</td>
<td>–</td>
<td>0.36</td>
</tr>
<tr>
<td>2 tabs</td>
<td>0.58</td>
<td>–</td>
<td>0.48</td>
</tr>
<tr>
<td>4 tabs</td>
<td>–</td>
<td>–</td>
<td>0.58</td>
</tr>
</tbody>
</table>

The flow along the top (N) surface of the CB for Config. bd (bottom flow) remained attached for all configurations. The random nature of the CB pressure readings in Fig. [H.20(b)] was suggestive of the transient nature of this flow regime. Although flow was separated along the top half of the OW, pressure distributions for \(x > 0.6D_o\) were similar; the ideal distribution showed that this section of the diffuser did not generate as much pressure recovery. Differences in pressures were observed for \(x < 0.6D_o\) since pressure gradients were observed in the attached portion of the duct.

For Config. bd (outside flow), the tuft locations identified in Tab. [H.4] did not correlate well with the pressure distribution features observed in Fig. [H.20(c)]. Results with 0- and 2-tabs showed decreased pressure along the CB that was characteristic of locally accelerating flow. The 4-tab distribution trended the same as Config. bc, so vortices must have maintained attached flow.

The OW distribution with 0 tabs had a decreasing pressure region that was indicative of a jet-flow and self-contained reversed core flow region. The OW pressure distributions with 2- and 4-tabs showed that the tabs created a diffusing effect. The discrepancy between SW and NE tap readings for \(x < 0.5D_o\) were indicative of locally different entrance effects, such as a slight misalignment between the transition duct and OW.
Outlet

Without swirl and tabs, contour plots for Config. bd (bottom flow) and (outside flow) are shown in Figs. G.1(b) and G.1(a) respectively. Figures H.21(a) and H.21(b) show the same configurations with four tabs. For the bottom flow regime, the top two tabs had no apparent impact on the flow development. The erroneous vectors pointing SE in the outside flow regime were indicators of separated regions: in addition to core separation, OW separation also occurred in the NW, SW, and SE quadrants. The OW separation may be attributed to the tabs that deflected the flow to the W, S, and E quadrants but the jet-type flow behaviour prevented any significant mixing between high- and low-momentum fluid.

Figure H.21: All OWs tab study Config. bd outlet total pressure contours and velocity vectors with S0 swirl and 4 tabs.

Performance

Figure H.22(a) plots back pressure and total pressure coefficients for the given cases. Trendlines for Configs. bc and bd (bottom flow) indicated worsening performance as tabs were added. All linear curves have goodness of fit greater than $r^2 > 95\%$. Configuration bd (outside flow) finally confirmed that tabs can recover pressure. With 4 tabs, $C_p$ improved 17% versus no tabs. A 42% improvement in $C_p$ was also obtained;
however, the 40% discrepancies between inlet and outlet mass flow rates led to questionable confidence in this number. Pressure penalties were calculated with 6 tabs, which substantiates a preferential amount of tab blockage $A_{tab}$.

![Figure H.22: All OWs tab study with S0 swirl performance.](image)

Figure H.22(b) plots the dynamic pressure coefficient for the three cases. Negligible improvements were observed for Config. bc whereas appreciable improvements were observed for both Config. bd flow regimes. Although the mass-flow bias was also present in this parameter, assuming a consistent absolute bias on all values, the corrected curves would trend the same way. It was apparent that for Config. bd (outside flow) with 4 tabs, the amount of kinetic energy recovered by vortex mixing was greater than the amount of total pressure lost due to the obstruction presence. Confirmation with literature regarding the requirement to limit the use of augmentation devices in wide-angled diffusers was achieved; however, the improvement to $C_b$ through using tabs in the $AR = 6.18$ diffuser was below the $C_b$ of the $AR = 2.73$ diffuser without tabs.
H.3.2 S20 Swirl

Inlet

Figure H.23 shows the averaged total and static pressure profiles with S20 swirl at section 1 for each outer wall without tabs. Pressure error bars were calculated uncertainties whereas the radial bars were attributed to completing the traverses manually. The resulting axial velocity profiles were independent of the outer wall geometry and were also unaffected by the downstream presence of tabs. Inlet flow area blockage was $A_{B,t} = 0.017–0.041$.

![Figure H.23: All OWs tab study with S20 swirl inlet pressure profiles. Tests without tabs, non-dimensionalized by the dynamic pressure.](image)

Wall

Figures H.24(a)–(d) show static wall pressure distributions using the different outer walls in tests with S20 swirl. The features in these figures were described in association with Figs. H.20(a)–(c). By increasing the number of tabs in Configs. ba and bb, Figs. H.24(a) and (b) show that wall pressure decreased within the diffuser but increased the pressure at section 1. The higher inlet pressure was associated to the tab-induced pressure hill.

The CB wool tufts were either characterized as attached, fluttering (indicating the onset of separation), or reversed during experiment. Table H.5 shows that CB separation occurred sooner for the larger area OWs. For OWd, separation always occurred at 0.13$D_o$, so the values in the table note the first tuft with reversed
flow. These locations compared well with the flat CB pressure distributions, which were typical in flow recirculation regions. For OWc, Fig. H.24(c) shows levelled-off CB distributions with no tabs at 0.13\(D_o\) and with 4 tabs at 0.48\(D_o\). The 0- and 2-tabs Config. bd CB distributions, as shown in Fig H.24(d), were level for \(x > 0.36D_o\).

**Table H.5:** All OWs tab study with S20 swirl onset of centre-body separation and OWd location of reversed flow (axial coordinate /\(D_o\)).

<table>
<thead>
<tr>
<th>outer wall</th>
<th>a</th>
<th>b</th>
<th>c</th>
<th>d</th>
</tr>
</thead>
<tbody>
<tr>
<td>0 tabs</td>
<td>–</td>
<td>0.65</td>
<td>0.13</td>
<td>0.65</td>
</tr>
<tr>
<td>4 tabs</td>
<td>–</td>
<td>0.65</td>
<td>0.36</td>
<td>0.36</td>
</tr>
</tbody>
</table>
Outlet

Figures [H.25(a)]–[c] show averaged outlet axial velocity profiles for the respective outer wall configurations with S20 swirl and tabs. For the 3 smaller OWs, more tabs increased the centreline velocity and reduced the outer wall boundary layer thickness. For OWd, the presence of tabs caused a significant amount of distortion that, at the very least, eliminated the jet-flow along the outer wall. The more uniform profiles obtained with tabs was responsible for higher kinetic energy recovery.

![Outlet Velocity Profiles](image)

**Figure H.25:** All OWs tab study with S20 swirl outlet axial velocity profiles.

The outlet data were satisfactory for OWs a–c: the difference in inlet to outlet mass-flows for Configs. ba and bb was < 6%, and for Config. bc was < 10%. The Config. bd outlet mass flow rate was 30–50% greater than \( \dot{m}_b \). Visual observations for OWd tests recorded reversed cores of \( \sim 0.5r_e \), so the shown profiles are erroneous—the dashed portion in Fig. [H.25(a)] should be blanked. The extra dashed profiles shown in Fig. [H.25(c)] were traverses completed with longer sampling times on the suspicion that errors were due to low frequency transience; the additional tests were not successful at capturing usable profiles. Fortunately, the last OW pressure tab obtained reliable values of \( P_{wall,e} \). Since \( (P_{wall,e} - \bar{P}_e) / \langle \dot{q}_t \rangle \approx 0.05 \) was consistent amongst repeated Config. bd experiments, there was some confidence associated with \( C_p \).
H.3. ALL OUTER WALLS EXPERIMENTS WITH TABS

Performance

Figure H.26 plots pressure recovery against tab blockage for the four outer walls with S20 swirl. Negative linear relationships occurred in the three smaller OW diffusers between the number of tabs and pressure recovery. OWa and OWb were acknowledged as not being wide enough to benefit from diffuser augmentation; however, OWc was near the design parameters for maximum pressure recovery at the prescribed area ratio \[1\] and the line of first appreciable stall \[4\]. Additionally, OWc was of similar configuration to that used by Wood \[119\], so tabs were expected to improve pressure recovery. An evaluation of other types of vortex generators in this annular diffuser would be beneficial to comment on the capability of tabs to improve pressure recovery or establish that swirling flow was detrimental to the augmentation device performance.

![Figure H.26: All OWs tab study with S20 swirl pressure recovery vs. tab blockage.](image)

For OWd, a 22% improvement in pressure recovery occurred with four tabs. Both the inlet wall pressure in Fig. H.24(d) and more uniform outlet velocity profile in Fig. H.25(c) confirmed that this diffuser was wide enough and benefitted from the augmentation devices. Unfortunately, it was too wide since the tab configurations attempted were not able to eliminate the reversed flow at the outlet plane; the smaller diffusers recovered more pressure.

Figure H.27 plots back pressure and total pressure recovery against area ratio. Owing to the mass flow discrepancy associated with OWd, \(C_{p0}\) was expected to be up to 10% lower if \(p_{0e}\) was corrected with the actual outlet velocity to give the same mass flow through the inlet. Maximum diffuser effectiveness were
\( \eta = 72\%, \ 80\%, \ 74\%, \ \text{and} \ 38\% \) for OWs a–d respectively.

Back pressure coefficient was a reliable value with uncertainty < 2% since it was not dependent on the outlet data. There was confidence that adding 4 tabs to the OWd diffuser increased \( C_b \) by 4.6% over no tabs. The curves showed that \( C_b \) increased up to \( AR = 2.73 \) and was substantially lower at \( AR = 6.18 \). Although referring to volutes in impeller applications, Japikse [189] projected that pressure recoveries of 0.80 may be achievable (already at 0.75) with \( \eta = 0.85-0.92 \) and will likely have \( 2.8 < AR < 4.1 \). Future diffuser studies should be limited to this area ratio range.