Aerodynamic Characteristics Of A Gas Turbine Exhaust Diffuser With An Accompanying Exhaust Collection System

2012

Bryan Bernier
University of Central Florida

Find similar works at: https://stars.library.ucf.edu/etd

University of Central Florida Libraries http://library.ucf.edu

Part of the Mechanical Engineering Commons

STARS Citation

https://stars.library.ucf.edu/etd/2373

This Masters Thesis (Open Access) is brought to you for free and open access by STARS. It has been accepted for inclusion in Electronic Theses and Dissertations by an authorized administrator of STARS. For more information, please contact lee.dotson@ucf.edu.
ABSTRACT

The effects of an industrial gas turbine’s Exhaust Collector Box (ECB) geometry on static pressure recovery and total pressure loss were investigated in this study experimentally and computationally. This study aims to further understand how exit boundary conditions affect the performance of a diffuser system as well as the accuracy of industry standard computational models. A design of experiments approach was taken using a Box-Behnken design method for investigating three geometric parameters of the ECB. In this investigation, the exhaust diffuser remained constant through each test, with only the ECB being varied. A system performance analysis was conducted for each geometry using the total pressure loss and static pressure recovery from the diffuser inlet to the ECB exit. Velocity and total pressure profiles obtained with a hotwire anemometer and Kiel probe at the exit of the diffuser and at the exit of the ECB are also presented in this study. A total of 13 different ECB geometries are investigated at a Reynolds number of 60,000. Results obtained from these experimental tests are used to investigate the accuracy of a 3-dimensional RANS with realizable k-ε turbulence model from the commercial software package Star-CCM+. The study confirms the existence of strong counter-rotating helical vortices within the ECB which significantly affect the flow within the diffuser. Evidence of a strong recirculation zone within the ECB was found to force separation within the exhaust diffuser which imposed a circumferentially asymmetric pressure field at the inlet of the diffuser. Increasing the ECB width proved to decrease the magnitude of this effect, increasing the diffuser protrusion reduced this effect to a lesser degree. The combined effect of increasing the ECB Length and Width increased the expansion area ratio, proving to increase the system pressure recovery
by as much as 19% over the nominal case. Additionally, the realizable k-ε turbulence model was able to accurately rank all 13 cases in order by performance; however the predicted magnitudes of the pressure recovery and total pressure loss were poor for the cases with strong vortices. For the large volume cases with weak vortices, the CFD was able to accurately represent the total pressure loss of the system within 5%.
ACKNOWLEDGMENTS

Research for this thesis was conducted at the Center for Advanced Turbine and Energy Research, a Laboratory for Gas Turbine Heat Transfer, Aerodynamics, and Durability at the University of Central Florida’s Siemens Energy Center under the guidance of Dr. Jayanta S. Kapat. I would like to officially thank my professor and mentor Dr. Kapat for providing me with the resources and knowledge necessary to complete this thesis and degree. I would not have been able to do this without him, and thanks to his assistance I have already started working at my dream job as a gas turbine engineer.

I would like to acknowledge my committee members as well, Dr. Kapat, Dr. Raghavan and Dr. Deng for their time and their advice on this thesis.

A special thanks goes out to all of my coworkers at the C.A.T.E.R lab: Mark Ricklick (for his tough-love management style), Matt Golsen (for keeping me in shape with all of the ridiculous training episodes), Roberto Claretti (for the most infectious laugh on the face of the earth), Anthony Bravato (for fixing everything I broke), Michelle Valentino (for just being a good friend) and Brandon Ealy / Zack Little for helping run some experiments while I was out of town.

Furthermore, this thesis could not have been completed without the funding and experience gained from the Laboratory for Turbine Heat Transfer, Aerodynamics, and Durability at the Center for Advanced Turbine and Energy Research at the University of Central Florida.

Last but not least, I must thank my family for their endless support. Even through the worst of times, my family has always been behind me. I truly have the best family
anyone could have hoped for, and I wouldn’t change a thing about them even if I could.

I love you Mom and Dad.
# TABLE OF CONTENTS

LIST OF FIGURES .................................................................................................................. xi

LIST OF TABLES ...................................................................................................................... xxviii

LIST OF ABBREVIATIONS ........................................................................................................ xxix

CHAPTER 1: INTRODUCTION TO GAS TURBINES AND EXHAUST DIFFUSERS ..... 2

Brayton Cycles .......................................................................................................................... 2

Recuperated Brayton Cycle ....................................................................................................... 4

Combined Cycle ....................................................................................................................... 5

Cycle Comparisons .................................................................................................................. 7

Exhaust Diffusers .................................................................................................................... 9

Literature Review ................................................................................................................... 11

CHAPTER 2: EXPERIMENTAL OVERVIEW (METHODS AND MATERIALS) .......... 18

Flow Conditioning ..................................................................................................................... 18

Test Section ............................................................................................................................. 25

Actual Parts ............................................................................................................................. 36

Geometry Definition ................................................................................................................. 39

Data Acquisition ....................................................................................................................... 46

Data Reduction Procedure ...................................................................................................... 46

Uncertainty ............................................................................................................................... 48

Experimental Validation .......................................................................................................... 50
CHAPTER 6: FURTHER STUDIES AND RECOMMENDATIONS FROM LESSONS LEARNED .................................................................................................................. 247

Splash plates ........................................................................................................ 247

PVC Ducts ............................................................................................................. 250

Rapid Prototyped Pressure Taps ............................................................................ 252

Internal Vein Analysis ............................................................................................. 253

Surface Roughness Analysis .................................................................................. 255

Exhaust Collector Box ............................................................................................ 255

Possible Improvements .......................................................................................... 257

APPENDIX A: UNCERTAINTY ANALYSIS ............................................................... 259

APPENDIX B: EXCEL SPREADSHEETS ................................................................. 265

APPENDIX C: PUBLICATIONS .............................................................................. 267

Conference Publications ......................................................................................... 268

REFERENCES ......................................................................................................... 269
LIST OF FIGURES

Figure 1 - Ideal brayton cycle ................................................................. 2
Figure 2 - Non-ideal brayton cycle .......................................................... 3
Figure 3 - Recuperated brayton cycle schematic (top) at low pressure ratio (bottom-left) and high pressure ratio (bottom-right) ................................................................. 4
Figure 4 - Compiled chart of in service gas turbines in the form of efficiency vs power output (blue is recuperated, black is simple cycle, red is combined cycle) ..................... 8
Figure 5 - Compiled chart of in service gas turbines in the form of efficiency vs pressure ratio .................................................................................................................. 9
Figure 6 - Variations in the Industrial gas turbine exhaust diffuser collector box. SGT-700 (Gas, Steam & Hydro Turbines for Power) (Top Left), SGT-600 (EngineerDir) (Top Right), Solar Turbines (Hi-Tek Manufacturing) (Bottom Left), Solar-Mars 90 Gas Turbine (Solar Turbines)(Bottom Right) ............................................................................. 10
Figure 7 - Schematic of the flow distribution plenum ..................................... 19
Figure 8 - Preassembly of the screens and honeycombs .................................. 20
Figure 9 - Close up view of the honeycomb cores ......................................... 20
Figure 10 - Side view of the plenum ............................................................... 22
Figure 11 - Cross section showing measurement locations (every 30°) for the nozzle exit. The dotted line indicating the inner annulus depicts where the inner annulus would be if it was connected .................................................................................................. 22
Figure 12 - Axial cross section showing measurement locations at the exit of the diffuser ......................................................................................................................... 23
Figure 13 - Radial cross section showing measurement locations at the exit of the diffuser ................................................................. 24
Figure 14 - Circumferential gage total pressure variations at the exit of the nozzle using a pegboard ............................................................... 24
Figure 15 Circumferential gage total pressure variations at the exit of the diffuser using a pegboard ............................................................................. 25
Figure 16 - Outer Nozzle and Inner Nozzle. Flow direction is left to right ......... 27
Figure 17 - Cross section of the nozzle showing the location of the optional swirler..... 29
Figure 18 - Cross Section view of the flow path of the diffuser ................................................. 31
Figure 19 - Static pressure port locations along the diffuser outer annulus .............. 32
Figure 20 - Exploded view of the nozzle/diffuser assembly. From left to right: Outer Nozzle, Inner Nozzle, Optional Outer Swirler, Optional Inner Swirler, Diffuser ............. 33
Figure 21 - Actual photo of an ECB geometry lined with a flexible Lexan sheet .......... 34
Figure 22 - Schematic of the pressure tap rows on the ECB ............................................. 35
Figure 23 - Actual photo of a completed ECB ........................................................................ 35
Figure 24 - Actual photo of the outer nozzle (white) and inner nozzle (blue) with one section left out for viewing angle ......................................................... 36
Figure 25 - Actual photo of the outer nozzle (white) and the optional swirler piece (blue/gray) along with the inner nozzle on the side (blue). All of these are placed on the plenum exit ............................................................................................................. 37
Figure 26 - Actual Diffuser (currently in post processing phase in this picture). The open section is for the optical window ............................................................................. 38
Figure 27 - Entrance side of the diffuser (in post processing stage) ......................... 39
Figure 28 - Definition of the ECB Length, Width, and Protrusion ........................................ 40
Figure 29 - CAD image of Case 12 ................................................................................... 42
Figure 30 - CAD images for Case 1 through Case 5 and Case 13................................. 44
Figure 31 - CAD images of Case 6 through Case 11 .................................................... 45
Figure 32 - Scanivalve calibration curve........................................................................... 47
Figure 33 - Smooth pipe validation of the experimental setup. Static pressure normalized by the dynamic head (using bulk velocity, Vb) versus length normalized by the pipe diameter (Dp) ........................................................................... 51
Figure 34 - Example of the effect of normalizing Cp using the expanding wall in a diffuser .................................................................................................................. 54
Figure 35 - Cp Normalization comparison ..................................................................... 55
Figure 36 - Full experiment model used for the CFD inlet boundary condition .......... 60
Figure 37 - CFD inlet boundary condition for the diffuser ........................................... 62
Figure 38 - Pressure contours (left) and Velocity vectors (right) for the inlet conditioning CFD model ........................................................................................................ 63
Figure 39 - CFD model boundary conditions ................................................................. 65
Figure 40 - Grid Dependence Study .............................................................................. 66
Figure 41 - CFD model mesh ......................................................................................... 68
Figure 42 - CFD model mesh looking at the struts of the diffuser from within the air volume ....................................................................................................................... 69
Figure 43 - CFD model mesh of the diffuser inlet .......................................................... 69
Figure 44 - Diffuser inlet profiles for the diffuser with no ECB (Free Exhaust) ............ 70
Figure 45 - Pressure recovery along the outer annulus of the diffuser with no ECB attached ................................................................. 70

Figure 46 - Experimental and computational results for the velocity profile with no ECB from the ID \([(r-R_i)/(R_o-R_i)=0]\) to OD \([(r-R_i)/(R_o-R_i)=1]\) at the exit between the struts (top), at the exit behind the struts (mid) and at the diffuser inlet (bottom). Velocity is on the left axis, and turbulence intensity ratio is on the right axis........................................ 71

Figure 47 - Axial velocity profile at the exit of the diffuser under a free exit condition showing the asymmetric separation zones.......................................................................................... 72

Figure 48 - Circumferential static pressure at the inlet of the diffuser for the experimental Case 1........................................................................................................................................ 73

Figure 49 - Diffuser pressure recovery on the OD for the experimental Case 1 ........... 73

Figure 50 - ECB wall pressures on the diffuser's side for the experimental Case 1 ...... 74

Figure 51 - ECB wall pressures on the impingement side for the experimental Case 1 74

Figure 52 - Circumferential static pressure at the inlet of the diffuser for the experimental Case 2........................................................................................................................................ 75

Figure 53 - Diffuser pressure recovery on the OD for the experimental Case 2 ........... 75

Figure 54 - ECB wall pressures on the diffuser's side for the experimental Case 2 ...... 76

Figure 55 - ECB wall pressures on the impingement side for the experimental Case 2 76

Figure 56 - Circumferential static pressure at the inlet of the diffuser for the experimental Case 3........................................................................................................................................ 77

Figure 57 - Diffuser pressure recovery on the OD for the experimental Case 3 ........... 77

Figure 58 - ECB wall pressures on the diffuser's side for the experimental Case 3 ...... 78

Figure 59 - ECB wall pressures on the impingement side for the experimental Case 3 78
Figure 60 - Circumferential static pressure at the inlet of the diffuser for the experimental Case 4 .......................................................... 79
Figure 61 - Diffuser pressure recovery on the OD for the experimental Case 4 ............ 79
Figure 62 - ECB wall pressures on the diffuser's side for the experimental Case 4 ...... 80
Figure 63 - ECB wall pressures on the impingement side for the experimental Case 4 80
Figure 64 - Circumferential static pressure at the inlet of the diffuser for the experimental Case 5 ........................................................................................................... 81
Figure 65 - Diffuser pressure recovery on the OD for the experimental Case 5 .......... 81
Figure 66 - ECB wall pressures on the diffuser's side for the experimental Case 5 ...... 82
Figure 67 - ECB wall pressures on the impingement side for the experimental Case 5 82
Figure 68 - Circumferential static pressure at the inlet of the diffuser for the experimental Case 6 ........................................................................................................... 83
Figure 69 - Diffuser pressure recovery on the OD for the experimental Case 6 ............ 83
Figure 70 - ECB wall pressures on the diffuser's side for the experimental Case 6 ...... 84
Figure 71 - ECB wall pressures on the impingement side for the experimental Case 6 84
Figure 72 - Circumferential static pressure at the inlet of the diffuser for the experimental Case 7 ........................................................................................................... 85
Figure 73 - Diffuser pressure recovery on the OD for the experimental Case 7 ............ 85
Figure 74 - ECB wall pressures on the diffuser's side for the experimental Case 7 ...... 86
Figure 75 - ECB wall pressures on the impingement side for the experimental Case 7 86
Figure 76 - Circumferential static pressure at the inlet of the diffuser for the experimental Case 8 ........................................................................................................... 87
Figure 77 - Diffuser pressure recovery on the OD for the experimental Case 8 ............ 87
Figure 78 - ECB wall pressures on the diffuser’s side for the experimental Case 8 ...... 88
Figure 79 - ECB wall pressures on the impingement side for the experimental Case 8 88
Figure 80 - Circumferential static pressure at the inlet of the diffuser for the experimental
Case 9............................................................................................................................................... 89
Figure 81 - Diffuser pressure recovery on the OD for the experimental Case 9............. 89
Figure 82- ECB wall pressures on the diffuser's side for the experimental Case 9 ...... 90
Figure 83 - ECB wall pressures on the impingement side for the experimental Case 9 90
Figure 84 - Circumferential static pressure at the inlet of the diffuser for the experimental
Case 10............................................................................................................................................... 91
Figure 85 - Diffuser pressure recovery on the OD for the experimental Case 10......... 91
Figure 86 - ECB wall pressures on the diffuser's side for the experimental Case 10 .... 92
Figure 87 - ECB wall pressures on the impingement side for the experimental Case 10
.......................................................................................................................................................... 92
Figure 88 - Circumferential static pressure at the inlet of the diffuser for the experimental
Case 11............................................................................................................................................... 93
Figure 89 - Diffuser pressure recovery on the OD for the experimental Case 11........ 93
Figure 90 - ECB wall pressures on the diffuser's side for the experimental Case 11 .... 94
Figure 91 - ECB wall pressures on the impingement side for the experimental Case 11
.......................................................................................................................................................... 94
Figure 92 - Circumferential static pressure at the inlet of the diffuser for the experimental
Case 12............................................................................................................................................... 95
Figure 93 -Diffuser pressure recovery on the OD for the experimental Case 12......... 95
Figure 94 -ECB wall pressures on the diffuser's side for the experimental Case 12 ..... 96
Figure 95 - ECB wall pressures on the impingement side for the experimental Case 12
........................................................................................................................................96
Figure 96 - Circumferential static pressure at the inlet of the diffuser for the experimental Case 13..........................................................................................................................................................97
Figure 97 - Diffuser pressure recovery on the OD for the experimental Case 13........... 97
Figure 98 - ECB wall pressures on the diffuser's side for the experimental Case 13 .... 98
Figure 99 - ECB wall pressures on the impingement side for the experimental Case 13
......................................................................................................................................................... 98
Figure 100 - Case 1 Geometry................................................................................................. 99
Figure 101 - Diffuser pressure recovery on the OD for the CFD Case 1 ....................... 99
Figure 102 - ECB wall pressures on the diffuser's side for the CFD Case 1 .............. 100
Figure 103 - ECB wall pressures on the impingement side for the CFD Case 1 ........ 100
Figure 104 - Symmetry plane velocity vectors for the CFD Case 1......................... 101
Figure 105 - Diffuser inlet and exit velocity vectors for the CFD Case 1 .................. 102
Figure 106 - Breakout plane velocity vectors for the CFD Case 1........................... 102
Figure 107 - ECB exit velocity vectors for the CFD Case 1...................................... 103
Figure 108 - Diffuser inlet and exit pressure profiles for the CFD Case 1 ............... 103
Figure 109 - Diffuser OD wall pressure for the CFD Case 1 ................................. 104
Figure 110 - ECB wall pressure on the diffuser side for the CFD Case 1............ 104
Figure 111 - ECB wall pressure for the impingement side for the CFD Case 1...... 105
Figure 112 - ECB exit total pressure profile for the CFD Case 1............................ 105
Figure 113 - Symmetry plane pressure profile for the CFD Case 1....................... 106
Figure 114 - Case 2 Geometry............................................................................................. 107
Figure 115 - Diffuser pressure recovery on the OD for the CFD Case 2

Figure 116 - ECB wall pressures on the diffuser's side for the CFD Case 2

Figure 117 - ECB wall pressures on the impingement side for the CFD Case 2

Figure 118 - Symmetry plane velocity vectors for the CFD Case 2

Figure 119 - Diffuser inlet and exit velocity vectors for the CFD Case 2

Figure 120 - Breakout plane velocity vectors for the CFD Case 2

Figure 121 - ECB exit velocity vectors for the CFD Case 2

Figure 122 - Diffuser inlet and exit pressure profiles for the CFD Case 2

Figure 123 - Diffuser OD wall pressure for the CFD Case 2

Figure 124 - ECB wall pressure on the diffuser side for the CFD Case 2

Figure 125 - ECB wall pressure for the impingement side for the CFD Case 2

Figure 126 - ECB exit total pressure profile for the CFD Case 2

Figure 127 - Symmetry plane pressure profile for the CFD Case 2

Figure 128 - Case 3 Geometry

Figure 129 - Diffuser pressure recovery on the OD for the CFD Case 3

Figure 130 - ECB wall pressures on the diffuser's side for the CFD Case 3

Figure 131 - ECB wall pressures on the impingement side for the CFD Case 3

Figure 132 - Symmetry plane velocity vectors for the CFD Case 3

Figure 133 - Diffuser inlet and exit velocity vectors for the CFD Case 3

Figure 134 - Breakout plane velocity vectors for the CFD Case 3

Figure 135 - ECB exit velocity vectors for the CFD Case 3

Figure 136 - Diffuser inlet and exit pressure profiles for the CFD Case 3

Figure 137 - Diffuser OD wall pressure for the CFD Case 3
Figure 138 - ECB wall pressure on the diffuser side for the CFD Case 3 ......................... 120
Figure 139 - ECB wall pressure for the impingement side for the CFD Case 3 ............... 121
Figure 140 - ECB exit total pressure profile for the CFD Case 3 .................................. 121
Figure 141 - Symmetry plane pressure profile for the CFD Case 3 ............................... 122
Figure 142 - Case 4 Geometry ...................................................................................... 123
Figure 143 - Diffuser pressure recovery on the OD for the CFD Case 4 ....................... 123
Figure 144 - ECB wall pressures on the diffuser's side for the CFD Case 4 ............... 124
Figure 145 - ECB wall pressures on the impingement side for the CFD Case 4 ......... 124
Figure 146 - Symmetry plane velocity vectors for the CFD Case 4 ............................ 125
Figure 147 - Diffuser inlet and exit velocity vectors for the CFD Case 4 ................. 126
Figure 148 - Breakout plane velocity vectors for the CFD Case 4 ............................. 126
Figure 149 - ECB exit velocity vectors for the CFD Case 4 ........................................ 127
Figure 150 - Diffuser inlet and exit pressure profiles for the CFD Case 4 ............... 127
Figure 151 - Diffuser OD wall pressure for the CFD Case 4 ........................................ 128
Figure 152 - ECB wall pressure on the diffuser side for the CFD Case 4 ................... 128
Figure 153 - ECB wall pressure for the impingement side for the CFD Case 4 ....... 129
Figure 154 - ECB exit total pressure profile for the CFD Case 4 ............................... 129
Figure 155 - Symmetry plane pressure profile for the CFD Case 4 ............................. 130
Figure 156 - Case 5 Geometry ...................................................................................... 131
Figure 157 - Diffuser pressure recovery on the OD for the CFD Case 5 .................... 131
Figure 158 - ECB wall pressures on the diffuser's side for the CFD Case 5 ............ 132
Figure 159 - ECB wall pressures on the impingement side for the CFD Case 5 ...... 132
Figure 160 - Symmetry plane velocity vectors for the CFD Case 5 .......................... 133
Figure 161 - Diffuser inlet and exit velocity vectors for the CFD Case 5 ..................... 134
Figure 162 - Breakout plane velocity vectors for the CFD Case 5 ......................... 134
Figure 163 - ECB exit velocity vectors for the CFD Case 5 .................................. 135
Figure 164 - Diffuser inlet and exit pressure profiles for the CFD Case 5 .............. 135
Figure 165 - Diffuser OD wall pressure for the CFD Case 5 ................................. 136
Figure 166 - ECB wall pressure on the diffuser side for the CFD Case 5 ............. 136
Figure 167 - ECB wall pressure for the impingement side for the CFD Case 5 ...... 137
Figure 168 - ECB exit total pressure profile for the CFD Case 5 ....................... 137
Figure 169 - Symmetry plane pressure profile for the CFD Case 5 .................... 138
Figure 170 - Case 6 Geometry ............................................................................... 139
Figure 171 - Diffuser pressure recovery on the OD for the CFD Case 6 ............. 139
Figure 172 - ECB wall pressures on the diffuser's side for the CFD Case 6 ....... 140
Figure 173 - ECB wall pressures on the impingement side for the CFD Case 6 ...... 140
Figure 174 - Symmetry plane velocity vectors for the CFD Case 6 .................... 141
Figure 175 - Diffuser inlet and exit velocity vectors for the CFD Case 6 .......... 142
Figure 176 - Breakout plane velocity vectors for the CFD Case 6 ...................... 142
Figure 177 - ECB exit velocity vectors for the CFD Case 6 ................................. 143
Figure 178 - Diffuser inlet and exit pressure profiles for the CFD Case 6 .......... 143
Figure 179 - Diffuser OD wall pressure for the CFD Case 6 ................................. 144
Figure 180 - ECB wall pressure on the diffuser side for the CFD Case 6 .......... 144
Figure 181 - ECB wall pressure for the impingement side for the CFD Case 6 .... 145
Figure 182 - ECB exit total pressure profile for the CFD Case 6 ....................... 145
Figure 183 - Symmetry plane pressure profile for the CFD Case 6 .................... 146
Figure 184 - Case 7 Geometry

Figure 185 - Diffuser pressure recovery on the OD for the CFD Case 7

Figure 186 - ECB wall pressures on the diffuser's side for the CFD Case 7

Figure 187 - ECB wall pressures on the impingement side for the CFD Case 7

Figure 188 - Symmetry plane velocity vectors for the CFD Case 7

Figure 189 - Diffuser inlet and exit velocity vectors for the CFD Case 7

Figure 190 - Breakout plane velocity vectors for the CFD Case 7

Figure 191 - ECB exit velocity vectors for the CFD Case 7

Figure 192 - Diffuser inlet and exit pressure profiles for the CFD Case 7

Figure 193 - Diffuser OD wall pressure for the CFD Case 7

Figure 194 - ECB wall pressure on the diffuser side for the CFD Case 7

Figure 195 - ECB wall pressure for the impingement side for the CFD Case 7

Figure 196 - ECB exit total pressure profile for the CFD Case 7

Figure 197 - Symmetry plane pressure profile for the CFD Case 7

Figure 198 - Case 8 Geometry

Figure 199 - Diffuser pressure recovery on the OD for the CFD Case 8

Figure 200 - ECB wall pressures on the diffuser's side for the CFD Case 8

Figure 201 - ECB wall pressures on the impingement side for the CFD Case 8

Figure 202 - Symmetry plane velocity vectors for the CFD Case 8

Figure 203 - Diffuser inlet and exit velocity vectors for the CFD Case 8

Figure 204 - Breakout plane velocity vectors for the CFD Case 8

Figure 205 - ECB exit velocity vectors for the CFD Case 8

Figure 206 - Diffuser inlet and exit pressure profiles for the CFD Case 8
Figure 207 - Diffuser OD wall pressure for the CFD Case 8 .................................................. 160
Figure 208 - ECB wall pressure on the diffuser side for the CFD Case 8......................... 160
Figure 209 - Rescaled image of the ECB wall pressure on the diffuser side for the CFD Case 8 ........................................................................................................................................... 161
Figure 210 - ECB wall pressure for the impingement side for the CFD Case 8.............. 161
Figure 211 - Rescaled image of the ECB wall pressure for the impingement side for the CFD Case 8 ........................................................................................................................................... 162
Figure 212 - ECB exit total pressure profile for the CFD Case 8...................................... 162
Figure 213 - Symmetry plane pressure profile for the CFD Case 8................................. 163
Figure 214 - Case 9 Geometry .............................................................................................. 164
Figure 215 - Diffuser pressure recovery on the OD for the CFD Case 9....................... 164
Figure 216 - ECB wall pressures on the diffuser's side for the CFD Case 9 ............... 165
Figure 217 - ECB wall pressures on the impingement side for the CFD Case 9........... 165
Figure 218 - Symmetry plane velocity vectors for the CFD Case 9............................... 166
Figure 219 - Diffuser inlet and exit velocity vectors for the CFD Case 9 ..................... 167
Figure 220 - Breakout plane velocity vectors for the CFD Case 9................................. 167
Figure 221 - ECB exit velocity vectors for the CFD Case 9.............................................. 168
Figure 222 - Diffuser inlet and exit pressure profiles for the CFD Case 9 ..................... 168
Figure 223 - Diffuser OD wall pressure for the CFD Case 9 ........................................... 169
Figure 224 - ECB wall pressure on the diffuser side for the CFD Case 9.................... 169
Figure 225 - Rescaled image of the ECB wall pressure on the diffuser side for the CFD Case 9 ........................................................................................................................................... 170
Figure 226 - ECB wall pressure for the impingement side for the CFD Case 9........... 170
Figure 227 - Rescaled image of the ECB wall pressure for the impingement side for the CFD Case 9 ................................................................. 171
Figure 228 - ECB exit total pressure profile for the CFD Case 9 ...................... 171
Figure 229 - Symmetry plane pressure profile for the CFD Case 9 .................... 172
Figure 230 - Case 10 Geometry ..................................................................... 173
Figure 231 - Diffuser pressure recovery on the OD for the CFD Case 10 ............ 173
Figure 232 - ECB wall pressures on the diffuser's side for the CFD Case 10 .......... 174
Figure 233 - ECB wall pressures on the impingement side for the CFD Case 10 .... 174
Figure 234 - Symmetry plane velocity vectors for the CFD Case 10 .................... 175
Figure 235 - Diffuser inlet and exit velocity vectors for the CFD Case 10 ............ 176
Figure 236 - Breakout plane velocity vectors for the CFD Case 10 .................... 176
Figure 237 - ECB exit velocity vectors for the CFD Case 10 ............................ 177
Figure 238 - Diffuser inlet and exit pressure profiles for the CFD Case 10 .......... 177
Figure 239 - Diffuser OD wall pressure for the CFD Case 10 ............................ 178
Figure 240 - ECB wall pressure on the diffuser side for the CFD Case 10 .......... 178
Figure 241 - ECB wall pressure for the impingement side for the CFD Case 10 .... 179
Figure 242 - ECB exit total pressure profile for the CFD Case 10 ...................... 179
Figure 243 - Symmetry plane pressure profile for the CFD Case 10 .................... 180
Figure 244 - Case 11 Geometry ..................................................................... 181
Figure 245 - Diffuser pressure recovery on the OD for the CFD Case 11 .......... 181
Figure 246 - ECB wall pressures on the diffuser's side for the CFD Case 11 .......... 182
Figure 247 - ECB wall pressures on the impingement side for the CFD Case 11 .... 182
Figure 248 - Symmetry plane velocity vectors for the CFD Case 11 .................... 183
Figure 249 - Diffuser inlet and exit velocity vectors for the CFD Case 11 ...................... 184
Figure 250 - Breakout plane velocity vectors for the CFD Case 11......................... 184
Figure 251 - ECB exit velocity vectors for the CFD Case 11................................ 185
Figure 252 - Diffuser inlet and exit pressure profiles for the CFD Case 11 .......... 185
Figure 253 - Diffuser OD wall pressure for the CFD Case 11 ............................ 186
Figure 254 - ECB wall pressure on the diffuser side for the CFD Case 11 ............ 186
Figure 255 - ECB wall pressure for the impingement side for the CFD Case 11 .... 187
Figure 256 - ECB exit total pressure profile for the CFD Case 11...................... 187
Figure 257 - Symmetry plane pressure profile for the CFD Case 11.................... 188
Figure 258 - Case 12 Geometry........................................................................ 189
Figure 259 - Diffuser pressure recovery on the OD for the CFD Case 12 .......... 189
Figure 260 - ECB wall pressures on the diffuser's side for the CFD Case 12 ....... 190
Figure 261 - ECB wall pressures on the impingement side for the CFD Case 12 ...... 190
Figure 262 - Symmetry plane velocity vectors for the CFD Case 12................. 191
Figure 263 - Diffuser inlet and exit velocity vectors for the CFD Case 12 ............ 192
Figure 264 - Breakout plane velocity vectors for the CFD Case 12 ..................... 192
Figure 265 - ECB exit velocity vectors for the CFD Case 12.............................. 193
Figure 266 - Diffuser inlet and exit pressure profiles for the CFD Case 12 .......... 193
Figure 267 - Diffuser OD wall pressure for the CFD Case 12 ............................ 194
Figure 268 - ECB wall pressure on the diffuser side for the CFD Case 12 .......... 194
Figure 269 - ECB wall pressure for the impingement side for the CFD Case 12 .... 195
Figure 270 - ECB exit total pressure profile for the CFD Case 12 ...................... 195
Figure 271 - Symmetry plane pressure profile for the CFD Case 12................. 196
| Figure 272 | Case 13 Geometry | 197 |
| Figure 273 | Diffuser pressure recovery on the OD for the CFD Case 13 | 197 |
| Figure 274 | ECB wall pressures on the diffuser's side for the CFD Case 13 | 198 |
| Figure 275 | ECB wall pressures on the impingement side for the CFD Case 13 | 198 |
| Figure 276 | Symmetry plane velocity vectors for the CFD Case 13 | 199 |
| Figure 277 | Diffuser inlet and exit velocity vectors for the CFD Case 13 | 200 |
| Figure 278 | Breakout plane velocity vectors for the CFD Case 13 | 200 |
| Figure 279 | ECB exit velocity vectors for the CFD Case 13 | 201 |
| Figure 280 | Diffuser inlet and exit pressure profiles for the CFD Case 13 | 201 |
| Figure 281 | Diffuser OD wall pressure for the CFD Case 13 | 202 |
| Figure 282 | ECB wall pressure on the diffuser side for the CFD Case 13 | 202 |
| Figure 283 | ECB wall pressure for the impingement side for the CFD Case 13 | 203 |
| Figure 284 | ECB exit total pressure profile for the CFD Case 13 | 203 |
| Figure 285 | Symmetry plane pressure profile for the CFD Case 13 | 204 |
| Figure 286 | Experimental Box Behnken Response Surface for Inlet Pressure Variation | 205 |
| Figure 287 | Experimental Box Behnken Response Surface for Inlet Pressure Variation | 206 |
| Figure 288 | Experimental Box Behnken Response Surface for Inlet Pressure Variation | 206 |
| Figure 289 | Experimental Box Behnken Response Surface for System Cp | 207 |
| Figure 290 | Experimental Box Behnken Response Surface for System Cp | 207 |
| Figure 291 | Experimental Box Behnken Response Surface for System Cp | 208 |
Figure 292 - Experimental Box Behnken Response Surface for System Total Pressure Loss.............................................................................................................................................................................................. 208
Figure 293 - Experimental Box Behnken Response Surface for System Total Pressure Loss.............................................................................................................................................................................................................................................................. 209
Figure 294 - Experimental Box Behnken Response Surface for System Total Pressure Loss.............................................................................................................................................................................................................................................................. 209
Figure 295 - Computational Box Behnken Response Surface for System Cp............................ 213
Figure 296 - Computational Box Behnken Response Surface for System Cp............................ 213
Figure 297 - Computational Box Behnken Response Surface for System Cp............................ 214
Figure 298 - Computational Box Behnken Response Surface for System Total Pressure Loss.............................................................................................................................................................................................................................................................. 214
Figure 299 - Computational Box Behnken Response Surface for System Total Pressure Loss.............................................................................................................................................................................................................................................................. 215
Figure 300 - Computational Box Behnken Response Surface for System Total Pressure Loss.............................................................................................................................................................................................................................................................. 215
Figure 301 - Experimental System Performance Comparison ................................................. 227
Figure 302 - CFD System Performance Comparison................................................................. 232
Figure 303 - Area ratio definition between the Diffuser exit (A1 revolved around the Centerline) and the area between the Protrusion and the ECB wall (A2 revolved around the Centerline).............................................................................................................................................................................................................................................................. 236
Figure 304 - Experimental and Computational comparison of the system Cp............................ 240
Figure 305 - Experimental and Computational comparison of the system total pressure loss.............................................................................................................................................................................................................................................................................................................................. 241
Figure 306 - Side view schematic of the splash plate design ........................................ 248
Figure 307 - Top view schematic of the splash plate design ........................................ 248
Figure 308 - Nozzle exit total pressure variation with the splash plate design ............ 249
Figure 309 - Diffuser exit total pressure variation with the splash plate design ............ 249
Figure 310 - Discreet holes placed in PVC pipes at the bottom of the plenum .......... 251
Figure 311 - Gage total pressure variations circumferentially for the exit of the nozzle .................................................................................................................................................. 251
Figure 312 - Gage total pressure variations circumferentially for the exit of the diffuser .................................................................................................................................................. 252
Figure 313 - Internal pressure lines at the exit of the diffuser’s outer annulus ............ 253
LIST OF TABLES

Table 1 - Geometric parameters of the nozzle ................................................................. 28
Table 2 - Geometric parameters for the diffuser .............................................................. 30
Table 3 - Three variable Box-Behnken Design ............................................................... 41
Table 4 – Box-Behnken Design for this study ................................................................. 43
Table 5 - Transducer specifications and accuracy ............................................................ 49
Table 6 - Uncertainty analysis ......................................................................................... 49
Table 7 - Local average turbulence intensity in the diffuser ........................................... 72
Table 8 – Experimental Inlet Pressure Variation Coefficients ....................................... 210
Table 9 - Experimental System Cp Coefficients ............................................................ 211
Table 10 - Experimental System Total Pressure Loss Coefficients ............................... 212
Table 11 - CFD - System Cp Coefficients .................................................................... 216
Table 12 - CFD - System Total Pressure Loss Coefficients ......................................... 217
Table 13 – Experimental Inlet Cp Variation .................................................................. 224
LIST OF ABBREVIATIONS

English Symbols
D_h  Hydraulic Diameter
V_b  Bulk Velocity
D_p  Pipe Diameter
T_i  Turbulence Intensity
u    Axial velocity
v    Velocity in the y-direction
w    Velocity in the z-direction
C_p  Pressure Recovery Coefficient
P_s  Static Pressure
U    Fluid Velocity
X    Axial Distance
L    Diffuser Length
e    Relative Surface Roughness
R_o  Outer Radius
R_i  Inner Radius

Greek Symbols
u    Kinematic Viscosity
\rho Density

Subscripts
rms Root Mean Square
avg Average
0 or t Total

Superscripts
- Averaged Value

Abbreviations
CFD  Computational Fluid Dynamics
ECB  Exhaust Collector Box
GT   Gas Turbine
Re   Reynolds Number
RANS Reynolds Averaged Navier Stokes
SCFM Standard Cubic Feet per Minute
A typical gas turbine can be described in its simplest form using a Brayton cycle. All simple cycle gas turbines, regardless of their size or purpose, can be analyzed using the Brayton cycle shown in Figure 1.

Figure 1 - Ideal brayton cycle
Starting at state 1, representative of standard atmospheric conditions in most cases, the engine compresses the air isentropically (no change in entropy) from state 1 to state 2. This process may take place through a single stage centrifugal compressor, a multi stage axial compressor, or anything in between. From state 2 to state 3 is constant pressure heat addition, modeling the combustion chamber where fuel is added and burned. Once the combustion is complete, the compressed and heated air is passed through a turbine (state 3 to state 4) and expanded isentropically back to the original pressure.

The amount of work that can be theoretically extracted from the system is based on the change in enthalpy (h) from state 3 to state 4. However, in the vast majority of engines some of this power is used to drive the compressor. This enables the engine to be self-sustaining without the need of an external power source. Therefore, the net work output done by the engine is the work extracted by the turbine (h3 – h4) minus the work required by the compressor (h2 – h1). Due to the divergence of the constant pressure lines at increasing entropy, the turbine will always produce more power than the compressor requires (under the assumption of isentropic compression and expansion and heat addition at constant pressure). The efficiency of the engine can easily be calculated, as the ratio of the net work out to the work input to the system. In this case, the only work input to the system is through the addition of heat (i.e. fuel) through the combustion process.

The ideal Brayton cycle is a very simple and easy to use concept to quantify work outputs and efficiencies of a typical gas turbine. However, each component of a gas turbine is not performing at 100% efficiency, thus entropy is irreversibly added to the
system due to non-isentropic effects of compression, combustion, and expansion. These system losses can be seen in a more realistic Brayton cycle shown in Figure 2.

![Non-ideal Brayton cycle](image)

**Figure 2 - Non-ideal brayton cycle**

It is helpful to define the compressor efficiency as the ratio of the actual enthalpy change to the isentropic enthalpy change. More generally, this is the ratio of the change in energy of the working fluid (in most cases: air) to the energy required to drive the compressor. In a scenario where the compressor efficiency was 100%, all of the energy that drives the compressor is transferred to the working fluid. However, with a compressor efficiency of 0.9, only 90% of the energy required to drive the compressor is actually transferred to the fluid. This means that in order to achieve the same pressure ratio of an isentropic compressor, the non-ideal compressor requires 11.1% more energy input. Similarly, the combustor and turbine efficiencies can be defined as well. Typical values for compressor and turbine isentropic efficiencies are around 90%, while the combustor can reach efficiencies of over 98%.
Recuperated Brayton Cycle

There are two main ways to increase the thermal efficiency of an engine. Either increase the amount of work output from the system or reduce the amount of heat added to the system. The latter can be achieved by using what is called a recuperator. In many cases, the exhaust temperature is much higher than the ambient temperature and this leftover energy is wasted when the gas is discarded into the atmosphere. The idea behind a recuperator is to reclaim some of the heat that is exhausted into the atmosphere after the turbine (state 4 on the Brayton cycle).

Figure 3 - Recuperated brayton cycle schematic (top) at low pressure ratio (bottom-left) and high pressure ratio (bottom-right)
Through the use of a heat exchanger, the hot turbine exhaust gas (state 4) can be used to heat the compressor discharge air (state 2) prior to combustion. Assuming a perfect heat exchanger, this would increase the temperature from state 2 to state X, leaving only the distance between state X and state 3 required for heat input from the fuel. The net work output of the system is the same as a simple Brayton cycle (assuming no losses from the recuperator), however the heat input has decreased from (h3 – h2) to (h3 – hx), thus increasing the thermal efficiency. As you increase the pressure ratio of the engine, the temperature of state 2 also increases. There is a point at which the pressure ratio is high enough to bring the temperature at state 2 equal to state 4, at this point the recuperator can no longer exchange heat and is rendered useless (bottom right of Figure 3). It is because of this that recuperators are only used on low pressure ratio engines.

**Combined Cycle**

In order to increase the amount of work output from the system, a bottoming cycle can be added. Instead of using the high temperature turbine exhaust (state 4) for a recuperator (Figure 3), it can be sent into a vapor power cycle (Figure 4). This vapor cycle uses the exhaust heat from the turbine to heat high pressure steam to superheated temperatures, which is then passed through a vapor turbine which generates power.
Unlike the recuperated system, the vapor power cycle is independent of the pressure ratio of the gas turbine cycle (as long as the temperature at state 4 is approximately 600+ °C). For this reason, high pressure ratio engines use combined cycles rather than recuperated cycles.
Cycle Comparisons

A comparison was made between all publicly available operating conditions and efficiencies for in service gas turbines (simple cycle, recuperated, and combined cycle) from companies such as Alstom, Solar Turbines, Siemens, and various other companies.

On the bottom of Figure 5, the solid lines are non-ideal Brayton cycles with isentropic efficiencies of the compressor, turbine, and vapor (steam) turbine of 88%, thermal efficiency of the recuperator of 88% and a hot and cold side pressure loss of 2.5% each. Observations that can be made from this public information is that the very few (2) recuperated engines are at very low pressure ratios (less than 10). However they produce a higher efficiency than nearly all of the simple cycle engines. The combined cycle engines perform quite well, upwards of 60% efficiency, however they are unable to run at low pressure ratios due to the turbine exit temperature being too low to send into the vapor cycle. These low pressure ratio engines can theoretically achieve equivalently high efficiencies (>50%) at pressure ratios between 3 and 10 through the use of a recuperator.
Figure 4 - Compiled chart of in service gas turbines in the form of efficiency vs power output (blue is recuperated, black is simple cycle, red is combined cycle)
Exhaust Diffusers

The purpose of this gas turbine exhaust diffuser is to recover as much dynamic head as possible before the flow exits into the atmosphere, increasing the static pressure with minimum total pressure losses. This enables the turbine stage to experience a higher pressure ratio than it would if there was no exit diffuser, thus increasing the amount of work extracted by the turbine stages. The flow in this exhaust diffuser is still moving at a speed where the dynamic head is not negligible, thus a significant amount of static pressure could possibly be recovered through an efficient diffuser design. In the land based power generation turbines, or marine based propulsion turbines, the engine may be enclosed in a structure which requires proper ventilation of the exhaust gasses. The need for proper ventilation also occurs in vehicles such as tanks and helicopters. The
combination of the need for pressure recovery and the limited enclosed space is where the exhaust collector box (ECB) comes into the picture.

Figure 6 - Variations in the Industrial gas turbine exhaust diffuser collector box. SGT-700 (Gas, Steam & Hydro Turbines for Power) (Top Left), SGT-600 (EngineerDir) (Top Right), Solar Turbines (Hi-Tek Manufacturing) (Bottom Left), Solar-Mars 90 Gas Turbine (Solar Turbines)(Bottom Right)

In most cases, the exhaust gasses are directed upwards and out of the structure. The addition of this exhaust collector box is a necessary section in some turbines, however it does increase the total pressure loss of the system by adding additional ducting and turns to the flow path. The wide variation in industrial designs for the diffuser exhaust collector box design is depicted in the various designs in Figure 1. While every design is different, the ultimate goal of the exhaust system is equivalent in every engine. The fundamental aspects of diffuser performance (sensitive to inlet
profile, separation, etc.) are also common for every exhaust system. The goal of this study is to look at the flow structures within an exhaust system to gain some knowledge of what factors go into the pressure recovery and total pressure loss, and hopefully find ways to improve the performance. If the exhaust could be vented away in an efficient manner, the system performance could increase thus generating more power and reducing specific fuel consumption. In this study, we investigate the design of the exhaust collector box downstream of the engine’s final diffuser and how it affects the system performance.

**Literature Review**

Diffusers in general have been studied for many years, and the basic principles can be found in nearly every fluid mechanics textbook. For many years, these basic principles were used to design diffusers within the gas turbine with successful results. For many years (late-1960s to mid-1990s) the research on diffusers was almost nonexistent. This was due to the fact that research and inventions are driven by need, as you won’t try to solve a problem that doesn’t exist. If there are no problems with the diffusers that are being designed, than why spend the time and money to research them? In the mid-1990s, companies started to realize that the diffusers within their engines were not performing exactly as they expected. While they were still working, it became evident that the scientific community could not explain everything that was happening within the diffuser. This generated the problem, which in turn gave way to a series of research groups investigating the diffuser performance on a more detailed level. Within the last 15 years, the academic and industrial research groups have invested large sums of money into detailed experiments and computational models.
trying to predict and characterize the internals of a diffuser. The literature that is reviewed in this study focuses primarily on the recent work.

In 2007, a study done by (Mahalakshmi, Krithiga, Sandhya, Vikraman, & Ganesan, 2007) investigated two diffuser geometries (5 and 7 degree half-cone angles) with three velocity profiles at the inlet. All three velocity profiles were a radial profile, the first being a typical boundary layer profile (BL height of 20% of the radius at the inlet), the second being a “wake” profile using the wake of a streamlined body just upstream of the inlet, and the third was a wake profile using a bluff body just upstream of the inlet. A very detailed uncertainty analysis was performed and explained for the velocity and turbulence intensity levels for u, v, and w. Wake dissipation time within the diffuser behind the blunt and streamlined bodies are shown in this study. The wake behind the bluff body was a deep wake, which accelerated the flow around the edges of the diffuser, thus introducing more flow near the walls. The boundary layer growth along the walls was stunted due to this fact, creating a thinner boundary layer at the exit when compared to the no-wake and streamline wake cases. The wake dissipation happens much quicker in the larger diffusion angles, they claim this is because of the larger adverse pressure gradient. Relative Wake Depth (RWD) and wake half-width are investigated as well. The RWD is a measure of how deep the wake is compared to the free stream value, while the wake half-width is the radial distance covered by the wake (from maximum wake at centerline, to half of the difference between the max wake and no wake). For a streamline body, the RWD decreases nearly linearly for the 5 degree cone while the half-width increases just after the wake is initiated at the inlet, remains constant for a long period of time, then decreases near the end of the diffuser. The
RWD behind a bluff body is the same (generally) as the streamline body. The wake half-width still increases at the initial stages of the diffuser, however it increases nearly to \( X/L = 0.4 \) (while the streamline body only increases to \( X/L=0.2 \)) after which it decreases. When the diffuser is changed to the 7 degree half-cone, the RWD of the streamline body does not vary significantly, and remains nearly constant although the values are less than that of the 5 degree half-cone case. The half-width still increases in the initial portion, but remains constant for the remainder of the diffuser (does not decrease). The bluff body, on the other hand, shows a nearly linear decrease in the RWD throughout the length of the diffuser, and a constant half-width from about \( X/L=0.15 \) to 1. The overall pressure recovery of the 5 degree half cone angle increases slightly with the introduction of the wakes. The overall pressure recovery was not affected by the introduction of wake to the 7 degree half angle cone. The remaining portion of the paper goes into the \( u', v', w' \) turbulence intensities plotted for each case, at every downstream plane (13 planes from \( X/L=0 \) to 1) for each case. The turbulence intensities tend to vary quite a bit at the low \( X/L \) range, significantly more in the bluff wake case than the streamline case. Once \( X/L \) increases however, they tend to remain constant over the radius. The no-wake case has significantly higher turbulence in the boundary layer (to be expected). The 7 degree half-cone case had the turbulence intensity fluctuation changes die out to a constant value much faster than the 5 degree half-cone case.

For more detail on the turbulence within a conical diffuser, it is suggested that (Azad, 1996) be referenced. In this study, an extensive amount of turbulence data is presented for a typical conical diffuser from various probe types such as hot wire, pulsed wire, pitot tubes, local static pressure ports, as well as an empirical method.
Within this study, the data presented includes (but is not limited to): mean stress and strain rate, Reynolds stresses and higher moments, skewness and flatness factors, momentum, turbulent kinetic energy, shear stress equations, each term of the momentum equation, instantaneous flow reversals, velocity fluctuations and relative strength of the large eddies. This study concluded that pulsed wires were the most accurate experimental tool for capturing the turbulence effects within a diffuser, while the empirical method was also quite successful.

An annular diffuser was studied comparing the difference between an open diffuser, and a diffuser with struts by (Ubertini & Desideri, 2000). A scaled down model of an actual turbine exhaust diffuser (from the PGT10 gas turbine) was studied with an inlet speed around 80 m/s. The authors claim this is high enough to result in accurate Reynolds number comparisons even though they are at 6x10^5 and the engine is on the order of 10^6. Inlet guide vanes are placed at the inlet of the diffuser, and struts are placed within the diffuser (around halfway through at X/L=0.5). The diffuser used was an 8 degree expansion angle, with a total length of 450mm. Static pressure taps placed axially along, taken at the midplane between the struts, and one on the strut axis (the points on the strut are taken just along the surface of the strut as opposed to the centerline). Static pressure rises significantly in the stagnation region just in front of the strut, then decreases just afterwards. Along the length of the diffuser after the struts, both measurements were nearly equal. For the no-strut diffuser, the static pressure remained relatively constant when the radial position was changed, and showed very slight differences between the inlet guide vanes (measurements taken behind a vane, and at midplane between vanes). The strut diffuser showed a significant change in
static pressure rise when radial position changes. The shroud showed a significant
stagnation region just prior to the strut, while the hub showed a very slight stagnation
region. Both pressure fields converged after the struts however. The no-strut diffuser
performed significantly better than the strut diffuser. The conclusions from this study are
as follows: The diffusion in the duct with struts is interrupted by the reduction in area
due to the struts, so the flow has potentially more diffusion to achieve and thus higher
pressure gradients In the empty duct, a reduction of the pressure recovery gradient is
observed at the exit of the duct, due to separation. This is exacerbated when struts are
included in the diffuser. In both cases, efficiency calculated at the shroud is higher than
that at the hub; this is because the hub diameter is constant, while the shroud is
diffusing. Efficiency in the diffuser with struts is 10-15% lower than the empty diffuser.
Pressure recovery in empty diffuser is almost equal to an ideal one. Overall diffuser loss
is significantly increased by the struts and this loss rise mainly occurs in the axial region
of the struts and in the endwall regions, where flow is separating.

A scaled model of the GE-MS9001E gas turbine exhaust system was studied by
(Sultanian, Nagao, & Sakamoto, Experimental and Three-Dimensional CFD
Investigation in a Gas Turbine Exhaust System, 1999) both experimentally and
computationally under multiple loading conditions. Experimental data was taken in
between the struts, at strut outlet plane, and model outlet plane and compared with a 3-
dimensional RANS method with the standard high-Reynolds-number k-e turbulence
model. This study is similar to the present study, however the main focus of the former
was on the flow patterns within the diffuser with very little data on the exhaust stack. A
single geometry was studied while the loading condition (inlet velocity profile) was
varied. The total and static pressure losses were derived from the measurements, and were determined to be slightly higher than the computational results, concluding in the fact that the computation does not fully capture the secondary flow losses in the turning vanes. Overall, the computational model was a successful design tool which reasonably predicts the performance of a complex exhaust system. More recently, in 2010 a study was performed by (Hirschmann, Volkmer, Schatz, Finzel, Casey, & Montgomery, 2010) investigating the effect of the total pressure inlet profile on the performance of an axial diffuser. The geometry of their diffuser is similar to the present study, however it lacks the exhaust collector box. Their study also shows the weakness of the computational models, as they are inaccurately capturing the effect of separation and flow reversal.

The conclusions from this study are as follows:

- The flow field in the present exhaust system largely varies with gas turbine operating load conditions. The trends of such variation in total pressure loss and static pressure recovery as well as the local flow features are reasonably predicted by the three dimensional CFD calculations.

- In a quantitative comparison, CFD predictions are found to compare well with the experiments for strut surface pressure and strut outlet total pressure in the front part (diffuser section) of the model. At the model outlet plane, pressure levels are calculated somewhat higher than experiment, indicating that secondary flow losses in regions of turning vanes and the plenum are not fully captured by the turbulence model.

- Results indicate satisfactory prediction accuracy for total pressure loss and static pressure recovery in the diffuser section. For the entire exhaust system including
the plenum the CFD predictions of these values are found consistently somewhat less accurate under wide operating load variations from full speed no load (FSNL) to full speed full load (FSFL)

- Overall the applied CFD methods offers a useful design engineering tool capable of predicting complex gas turbines exhaust system flows including the quantitative prediction of the total pressure loss and static pressure recovery.

This study aims to look further into the flow structures within the exhaust collector box and why these structures have a detrimental effect on the diffuser performance.
CHAPTER 2: EXPERIMENTAL OVERVIEW (METHODS AND MATERIALS)

Flow Conditioning

The overall goal of the flow conditioning was to create a flat velocity profile at the inlet of the diffuser. Diffusers are incredibly sensitive to any variations in the inlet velocity profile, and the performance of the diffuser changes significantly when the inlet condition is changed. The easiest way to determine the performance of the diffuser and ECB system is to have a constant, repeatable, and predictable inlet condition. To obtain this, a series of flow conditioning devices were designed and tested to ensure an even velocity profile at the inlet. A large plenum was created upstream of the test section in order to dissipate any unwanted wake from the exit of the blower. The plenum acts as a large reservoir with a very low flow velocity, which is ideal for the introduction of flow conditioning devices which can further reduce turbulence. A flat board with small holes drilled in a matrix pattern through the entire board surface (peg board) was used to distribute the flow inside the plenum. The idea was that it would create a small pressure drop across the board, causing the incoming flow to pressurize the upstream section prior to the peg board. This will allow for a relatively even distribution through the holes placed in the board, regardless of the fact that the inlet of the flow is located on one side of the plenum. This design proved to perform very well, as it operated with a reasonably low pressure drop, and evenly distributed the flow throughout the plenum and into the diffuser.
Following this distribution system was a ¼” cell size 1.5” thick aluminum honeycomb used to remove any large eddies still remaining in the flow after the distribution system. Once the flow has traveled through the honeycomb, additional eddies will be created due to the exiting flow patterns, and may actually increase turbulence immediately after the honeycomb. This was alleviated by the introduction of screens downstream of the honeycomb. Two sections of screens are used in this design, and are spaced approximately 20% of the hydraulic diameter of the plenum apart*. These act as a flow conditioning to the exiting flow from the honeycomb, by breaking up any large eddies and only producing very small, short lived, eddies in return.

There are pressure drops through each stage of this flow conditioning, however due to the size of the plenum it was possible to maintain a significant portion of the pressure head from the blower. An even distribution of 500 CFM of airflow through the plenum only results in a velocity around 0.5 m/s.

Figure 7 - Schematic of the flow distribution plenum

*Note: The spacing percentage may vary depending on the specific design requirements and the flow characteristics.
Figure 8 - Preassembly of the screens and honeycombs

Figure 9 - Close up view of the honeycomb cores
Once the flow exits the plenum, it is contracted by an annular nozzle before it enters into the diffuser. The nozzle has an contraction ratio of 4.6 and is designed not only to accelerate the flow, but to compress the boundary layer as much as possible to give the diffuser inlet a relatively flat velocity profile at the entrance. The nozzle also acts as the final stage in the flow conditioning system, helping to remove any remaining effects from the final screen and to further reduce turbulence.

In order for us to measure the total pressure profile at the exit of the nozzle, the diffuser must be removed to give access to this section of the rig. Due to the connections between segments, the nozzle’s inner annulus must be removed as well. This leaves the outer section of the nozzle exiting into a straight pipe. The measurements were taken circumferentially at 30° increments at the mid-plane between the nozzle outer annulus and where the inner annulus would have been if it were attached. The measurements were taken using a hand held micro-manometer with a resolution of 0.1 Pa attached to a total pressure probe. The values were averaged over a span of 16 seconds at each location to reduce noise. These locations are shown below.
Figure 10 - Side view of the plenum

Figure 11 - Cross section showing measurement locations (every 30°) for the nozzle exit. The dotted line indicating the inner annulus depicts where the inner annulus would be if it was connected.
These results are compared with the total pressure profile at the diffuser exit under the same flow conditions, because the diffuser itself can exaggerate any fluctuations due to diffusion and possible separation. To complicate things further, the diffuser is supported structurally with six (6) symmetrical airfoils placed 60° apart circumferentially. The results reported have the data points split into areas that are directly downstream of an airfoil, and ones that are free of the airfoil wake. Radially, each measurement location is at the mid-plane between the outer annulus and the inner annulus, mimicking the locations at the diffuser inlet. These locations are shown in the figure below.

Figure 12 - Axial cross section showing measurement locations at the exit of the diffuser
Figure 13 - Radial cross section showing measurement locations at the exit of the diffuser

Figure 14 - Circumferential gage total pressure variations at the exit of the nozzle using a pegboard
Quantitatively, this design is very promising. At the nozzle exit, even at the maximum flow rate we can achieve (500 CFM), the pressure variations were on the order of the uncertainty of our measurement (+/- 1 Pa). However, maximum variation in pressure of 58% (max to min) at the exit of the diffuser is observed, resulting in a 25% velocity variation. This leads us to believe that the diffuser's performance is extremely sensitive to the inlet conditions. Even with a flat velocity profile at the inlet of the diffuser the exit profile is skewed due to the nature of the adverse pressure gradient, the effect of the airfoils, and slight variations in surface roughness which all disrupts the flow as it travels through the diffuser.

**Test Section**

With the ability to rapidly produce 3-D models that were designed in modeling software, we decided to create a complex design for the diffuser so that the measurement techniques would be much easier as the testing progressed. We created all of the internal pressure lines, pressure taps, flanges and connections in the model.
Nozzle (outer/inner)

Due to the nature of fluid mechanics, it is very beneficial to condition the flow at the lowest possible velocity. This causes minimal pressure loss through the honeycombs and screens, and enables us to reach a flat velocity profile at the inlet to our test section. However, this means that it is necessary to accelerate the flow just prior to the measurement section in order to reach the required Reynolds Number. Through the introduction of a nozzle, we are able to accelerate the flow significantly in a short axial distance, while also compressing the boundary layer so that we do not have a fully developed condition at our test section. It was desirable for us to obtain a flat velocity profile at the inlet to the diffuser, as well as mimicking external flow conditions (i.e. no fully developed conditions). This was done so that the inlet profile of the diffuser was not a variable and would remain constant and predictable throughout the testing schedule. The diffuser is extremely sensitive to the inlet conditions, and variations in the inlet will cause exponentially increased variations downstream causing discrepancies in the data obtained from the diffuser and the ECB. Our nozzle has a contraction ratio of 4.6, with a 3rd order polynomial area reduction where the slope of the wall at the inlet and exit are parallel with the free stream velocity vector (Eckert, Mort, & Jope, 1976). It is an annular nozzle, thus it has an inner and outer annulus area reduction that match geometrically, leaving a symmetrical nozzle.
The nozzle was fabricated by Mydea Technologies through a process called Fused Deposition Modeling (FDM). This is a relatively inexpensive process, especially for a piece of this size. In order to further reduce the cost, the interior of the walls are created with a honeycomb structure to reduce weight, and reduce cost. The penalty is that the structural stability of the piece is significantly weaker, however all structural concerns were accounted for by increasing the thickness of the walls.
Table 1 - Geometric parameters of the nozzle

<table>
<thead>
<tr>
<th>Geometric Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Axial Length</td>
<td>180 mm</td>
</tr>
<tr>
<td>Inlet Area</td>
<td>0.0577 m²</td>
</tr>
<tr>
<td>Exit Area</td>
<td>0.0125 m²</td>
</tr>
<tr>
<td>Area Ratio</td>
<td>4.60</td>
</tr>
<tr>
<td>Hydraulic Diameter (exit)</td>
<td>0.0557 m</td>
</tr>
</tbody>
</table>

Optional Swirler

With the anticipation of the introduction of a swirler to the flow path, we decided to design another section of the inlet which can allow for a swirler if it is deemed necessary in future phases. This section was designed to fit between the nozzle and the diffuser, and can easily be removed and replaced if necessary. This piece was also created using the FDM process from Mydea Technologies, however since the piece is much smaller than the nozzle the interior was not honeycombed. The current setup is simply a straight duct spacer between the exit of the nozzle and the inlet of the diffuser.
Now that the flow conditioning has been completed, the diffuser can begin the testing sections. The diffuser was modeled after a typical gas turbine exhaust diffuser. The major geometric features in are shown below. It should be noted that the effect of scaling this model down from actual engine size decreases the Reynolds and Mach numbers of the exhaust by nearly an order of magnitude. In order to accurately compare scaled tests, the similarity principle requires matching at least one (ideally both) of these values. Due to the fact that neither of these parameters have been matched, the trends are being used to validate the CFD model used in this study and will be scaled up within the CFD once all parts have been validated.
<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Axial Length (m)</td>
<td>.230</td>
</tr>
<tr>
<td>Inlet Area (m²)</td>
<td>.0125</td>
</tr>
<tr>
<td>Exit Area (m²)</td>
<td>.0353</td>
</tr>
<tr>
<td>Area Ratio</td>
<td>2.817</td>
</tr>
<tr>
<td>Hydraulic Diameter (m)</td>
<td>.0557</td>
</tr>
<tr>
<td>Diffusion Angle (non-Taper)</td>
<td>8°</td>
</tr>
<tr>
<td>Inlet Reynolds Number</td>
<td>63,900</td>
</tr>
<tr>
<td>Inlet Mach Number</td>
<td>.045</td>
</tr>
</tbody>
</table>

The diffuser was created by Mydea Technologies using a process called Objet Polyjet Modeling. This processes has a resolution of 0.0006" per layer, giving an extremely smooth surface. The inlet e/D of the Objet diffuser is 0.0002, which when combined with the operating Reynolds number of 63,900, gives a Darcy-Weisbach friction factor of 0.14 which is very close to a smooth wall. The diffuser has an area ratio of 2.8 and is outfitted with numerous pressure ports. There are pressure taps at 11 axial positions (spaced evenly from X/L=0 to X/L=1) at two circumferential positions (0° or 6:00 and 180° or 12:00) giving 22 ports evenly spaced around the diffuser.
The remaining half hemisphere was left open for flow visualization. Since the inlet and exit of the diffuser have pressure ports located at the edges, it was not possible to leave a straight pressure tap line for those locations. It was necessary to snake the pressure line internally through the walls of the diffuser to a more accessible point on the wall of the diffuser. The diffuser is ‘tapered’ at the inlet, leveling out to an 8° diffusion for the remainder of the axial length. The exit has a straight piece added to it (0° diffusion) so that it fits into the collector box correctly.
Figure 19 - Static pressure port locations along the diffuser outer annulus

Each piece was designed to snap together quickly and easily, with alternating flanges for ease of access and use. When assembled, each connection was sealed using silicone to ensure that no flow leaks in or out of the experiment. The exploded view of each part is shown below.
The exhaust collector box was created using an acrylic housing sized to fit the largest ECB geometry. It was outfitted with a tight fit optically clear lid which is pressed onto the top of the ECB with a gasket for sealing purposes. The ECB geometries themselves were cut using an in-house CNC machine (tolerances expected: +/-0.008") out of MDF. Due to the fact that MDF is prone to leakage when a pressure gradient is applied, the interior of the ECB was lined with a thin flexible Lexan sheet and sealed on each side with thin gasket tape. This enables the cheap MDF material to provide the structural support and tight tolerances, while the flexible acrylic liner and the acrylic housing provide a hydrodynamically smooth and leak-free surface for the air path. An image of a completed ECB geometry is shown in Figure 21.
Pressure taps were included on the ECB in order to capture the pressure field within the system. Four rows of pressure taps were included on both the bottom and top walls of the ECB (denoted as the diffuser-side, and impingement-side walls respectively). A total of 150+ pressure taps are located on the ECB, however due to the changing width of the geometries not all pressure taps were used in each test. Using the assumption of a geometric symmetry plane down the center of the ECB, only one half of the geometry was instrumented with pressure taps giving the ability to use a denser grid on the half of interest. A schematic of the locations chosen for these ECB tap rows (as well as the nomenclature of Row 1 to Row 4) is shown in
Figure 22 - Schematic of the pressure tap rows on the ECB

Figure 23 - Actual photo of a completed ECB
Actual Parts

During the manufacturing process, pictures were taken to ensure that the parts were being assembled correctly. Due to the size of the nozzle, it was split into 4 sections (90° arc each) and segmented together in post processing. The seams of these connections were sanded and smoothed prior to use. The complete inner nozzle is shown with ¾ of the outer nozzle in the following picture.

![Actual photo of the outer nozzle (white) and inner nozzle (blue) with one section left out for viewing angle](image)

Once the manufacturing was completed, the parts were assembled onto the rig and sealed together using gasket tape and silicone. This was done to ensure that there are no leaks in or out of the system which can cause discrepancies in the data obtained. The flanges were designed to be able to secure each piece together using standard ¼”
bolts. The assembled nozzle is shown below, with the inner nozzle standing next to it. The inner nozzle connects to the end of the diffuser and hangs inside the nozzle.

Figure 25 - Actual photo of the outer nozzle (white) and the optional swirler piece (blue/gray) along with the inner nozzle on the side (blue). All of these are placed on the plenum exit.

The diffuser was a much more complicated piece to build. Due to its size (0.45 m long including the extended inner annulus) it was necessary to split the build into various parts, and seal them together in post processing. The inner annulus is extended past the diffuser exit, as the inner annulus travels through the ECB as well. To complicate things further, during the Objet process support material is used inside the negative features of the part and are cleaned out during post processing. This meant that in order to clean out the internal pressure lines traveling through the diffuser must be split in half in order to have the support material removed. This, combined with the
excessive size for the Objet machine, caused the diffuser to be split into 15 separate sections which were then sealed back together. Each piece was sealed together with Bondo, and sanded smooth once completed. One section of the diffuser was designed to hold an optically clear window for flow visualization during a later phase of the project. This section was filled with a temporary window. The diffuser is shown below, the white paste is the remaining Bondo sealant which connects each of the 15 parts together.

Figure 26 - Actual Diffuser (currently in post processing phase in this picture). The open section is for the optical window
**Figure 27 - Entrance side of the diffuser (in post processing stage)**

**Geometry Definition**

Based on reviewing images of typical turbine exhaust stacks, as well as published literature (Bernier, Riclick, & Kapat, 2011), three main geometric variables are seen. These variables are the length and width of the ECB, as well as how far the diffuser protrudes into the ECB. With these three variables, it is possible to recreate most of the geometry seen in the gas turbine field, albeit simplified considerably. These variables are shown in Figure 28
In order to truly investigate how each of these variables interacts with each other with respect to the system performance, a full factorial design is preferred. This would mean choosing a few levels of each variable, and going through every possible combination. For example, if 3 levels of each variable were chosen (i.e. a small value, medium value, and large value) than a 3x3x3 matrix would be created giving 27 total experiments. This method of experimentation is the most robust and provides the best data for representing secondary and tertiary effects of variable changes in the geometry; unfortunately it also requires the most time and resources and was not within the scope of this study. Therefore, a reduced test matrix was created using the Box-Behnken Design Methodology. The idea behind the design of experiment approach using a Box-Behnken Design is to reduce the amount of experiments required to investigate how multiple factors (in this case geometric factors) affect the system performance of the exhaust diffuser. This process requires three levels for each variable (in this case: Length, Width, Protrusion) denoted by a +1, 0, and -1. Three factors with three levels in a Box-Behnken Design results in a total of 15 experiments, with 3 center
points (0,0,0). For this study specifically, the experimental and computational results for three identical geometries will be identical, therefore a total of 13 cases are required to complete the design. These 13 cases are shown in Table 3.

Table 3 - Three variable Box-Behnken Design

<table>
<thead>
<tr>
<th>Run</th>
<th>ECB Length</th>
<th>ECB Width</th>
<th>Diffuser Protrusion</th>
</tr>
</thead>
<tbody>
<tr>
<td>Case 1</td>
<td>0</td>
<td>-1</td>
<td>-1</td>
</tr>
<tr>
<td>Case 2</td>
<td>0</td>
<td>+1</td>
<td>-1</td>
</tr>
<tr>
<td>Case 3</td>
<td>0</td>
<td>+1</td>
<td>+1</td>
</tr>
<tr>
<td>Case 4</td>
<td>0</td>
<td>-1</td>
<td>+1</td>
</tr>
<tr>
<td>Case 5</td>
<td>-1</td>
<td>0</td>
<td>-1</td>
</tr>
<tr>
<td>Case 6</td>
<td>+1</td>
<td>0</td>
<td>-1</td>
</tr>
<tr>
<td>Case 7</td>
<td>+1</td>
<td>0</td>
<td>+1</td>
</tr>
<tr>
<td>Case 8</td>
<td>-1</td>
<td>0</td>
<td>+1</td>
</tr>
<tr>
<td>Case 9</td>
<td>-1</td>
<td>-1</td>
<td>0</td>
</tr>
<tr>
<td>Case 10</td>
<td>+1</td>
<td>-1</td>
<td>0</td>
</tr>
<tr>
<td>Case 11</td>
<td>+1</td>
<td>+1</td>
<td>0</td>
</tr>
<tr>
<td>Case 12</td>
<td>-1</td>
<td>+1</td>
<td>0</td>
</tr>
<tr>
<td>Case 13</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
</tbody>
</table>

At this point, the values for +1, 0, and -1 were chosen for each of the ECB geometries. These values were chosen based on prior literature (Bernier, Riclick, & Kapat, 2011) as well as industry experience. The range of these values was chosen to encompass
nearly every design that is seen in the field, with the goal of encompassing a global system performance rather than a smaller scale specific feature. The nominal value (0) for the ECB Length was chosen to be 1.8 times the diffuser hydraulic diameter (Dh) or more physically, 3.6 times the span of the last turbine stage. The nominal value for the ECB Width was chosen to be 8.2 times the hydraulic diameter, which in this study translates into 2.6 times the OD of the flow path at the exit of the turbine (inlet of the diffuser). The variants of these values (+1 and -1) were chosen to be +50% and -50% of the nominal value respectively. The third geometric variable, the Diffuser Protrusion, was chosen to be a function of the ECB Length ranging from 0% to 33% to 66% representing -1, 0, and +1 respectively. The adjusted design of experiments table is shown in Table 4.

![Figure 29 - CAD image of Case 12](image-url)
Table 4 – Box-Behnken Design for this study

<table>
<thead>
<tr>
<th>Run</th>
<th>ECB Length / Dh</th>
<th>ECB Width / Dh</th>
<th>Diffuser Protrusion / ECB Length</th>
</tr>
</thead>
<tbody>
<tr>
<td>Case 1</td>
<td>1.8</td>
<td>6.2</td>
<td>0%</td>
</tr>
<tr>
<td>Case 2</td>
<td>1.8</td>
<td>10.3</td>
<td>0%</td>
</tr>
<tr>
<td>Case 3</td>
<td>1.8</td>
<td>10.3</td>
<td>66%</td>
</tr>
<tr>
<td>Case 4</td>
<td>1.8</td>
<td>6.2</td>
<td>66%</td>
</tr>
<tr>
<td>Case 5</td>
<td>0.9</td>
<td>8.2</td>
<td>0%</td>
</tr>
<tr>
<td>Case 6</td>
<td>2.7</td>
<td>8.2</td>
<td>0%</td>
</tr>
<tr>
<td>Case 7</td>
<td>2.7</td>
<td>8.2</td>
<td>66%</td>
</tr>
<tr>
<td>Case 8</td>
<td>0.9</td>
<td>8.2</td>
<td>66%</td>
</tr>
<tr>
<td>Case 9</td>
<td>0.9</td>
<td>6.2</td>
<td>33%</td>
</tr>
<tr>
<td>Case 10</td>
<td>2.7</td>
<td>6.2</td>
<td>33%</td>
</tr>
<tr>
<td>Case 11</td>
<td>2.7</td>
<td>10.3</td>
<td>33%</td>
</tr>
<tr>
<td>Case 12</td>
<td>0.9</td>
<td>10.3</td>
<td>33%</td>
</tr>
<tr>
<td>Case 13</td>
<td>1.8</td>
<td>8.2</td>
<td>33%</td>
</tr>
</tbody>
</table>

The resulting geometries are shown in Figure 30, Figure 31, and Figure 29.
Figure 30 - CAD images for Case 1 through Case 5 and Case 13
Figure 31 - CAD images of Case 6 through Case 11
**Data Acquisition**

In order to determine the range of pressures obtained in the experiment, Siemens conducted a scale model CFD experiment to estimate the pressures within the diffuser and ECB. By comparing these results with a rough 1-D estimate (using Bernoulli’s Equation) it was possible to reasonably estimate the expected pressures. By looking at the exit plane of the diffuser, it was determined that the full range of pressures observed was on the order of $10^2$ Pa. In order to get an accurate distribution with a full scale range of $10^2$ Pa, it was necessary to be able to measure differences on the order of 1 Pa.

The majority of the data acquired during each test is done with the help of a Multiplex Scanivalve system. This system uses a single pressure transducer to read each port one at a time. Due to the extremely low pressures being measured in this experiment (on the order of 1 Pa) an external transducer with a full scale range (+/− 350 Pa) with a full scale accuracy of 0.25% (~0.9 Pa) was implemented into the Scanivalve system. The voltage output from this transducer is routed into a carrier demodulator, which is then fed into our data acquisition system. In this experiment, each port is read at a rate of 2 Hz for a total of 50 measurements. Once the port has finish recording the Scanivalve system switches to the next port and waits 2 seconds, giving the transducer enough time to adjust to its steady state value for that port.

**Data Reduction Procedure**

The raw data obtained from each test from the Scanivalve system is a raw voltage output from the transducer. Due to fluctuations in the flow field, 50 measurements were taken at each port (separated by 0.5 seconds each) and averaged
into a single value to reduce the amount of noise in the measurement (and to average out any short term periodicity). A full scale calibration was completed in house on the transducer using a U-tube manometer in a closed and controlled pressure system. The calibration curve is shown below. With this calibration curve, it is possible to relate the voltage output of the Scanivalve system to an actual gage pressure reading.

![Scanivalve Transducer Calibration](image)

**Figure 32 - Scanivalve calibration curve**

The pressure recovery coefficient ($C_p$) is a standard way to describe how effective a diffuser is a recovering pressure. $C_p$ is defined as follows:

$$C_p := \frac{P_2 - P_1}{\frac{1}{2} \rho V^2}$$

(1)

In this equation, $P_2$ is the pressure at each respective port in which $C_p$ is being calculated. $P_1$ is defined as the inlet static pressure of the diffuser which is defined later in the section titled Diffuser $C_p$ Normalization. The denominator is the total dynamic head of the flow, also taken at the inlet of the diffuser. Three total pressure probes are located at a single position in the entrance of the diffuser to measure the average total
pressure, and the velocity is calculated using Bernoulli’s Equation for subsonic incompressible flow shown below. The density is calculated using the ideal gas law, using the measured temperature (measured with a type T thermocouple) and the atmospheric pressure of that day.

\[
P_s + \frac{1}{2} \rho \cdot V^2 := P_t
\]

\[
V := \sqrt{\frac{2 \cdot (P_t - P_s)}{\rho}}
\]

(2)

The exhaust collector box is also defined using the same Cp as the diffuser, for consistency. This will enable us to tell how well the box is diffusing the flow relative to the diffuser exit, if the Cp value is higher in the ECB, than it is decreasing the local velocity, thus increasing \(P_2\). The coordinate system for the location of measurements on the ECB is presented in a later section.

**Uncertainty**

For this experiment, a basic 0th order uncertainty calculation was completed for the measurements taken in the tests. The biggest issue for this test was the extremely small pressures that we were expecting in the flow field. Due to the scaling of this rig, and the 500 CFM limitation on the blower, we were expecting the difference between the circumferential ports at the diffuser exit to be on the order of 1 Pa. With such a small variation in pressure, the transducers that we used must be able to work with a very low resolution and accuracy. In the table below, the transducers used in this experiment are shown along with their respective accuracies.
Table 5 - Transducer specifications and accuracy

<table>
<thead>
<tr>
<th>Transducer</th>
<th>Range (Pa)</th>
<th>Resolution/Accuracy</th>
</tr>
</thead>
<tbody>
<tr>
<td>Micro-Manometer</td>
<td>-750 to +548</td>
<td>0.1 Pa</td>
</tr>
<tr>
<td>Scanivalve Transducer</td>
<td>-350 to +350</td>
<td>+/- 0.25% FS (~0.9Pa)</td>
</tr>
</tbody>
</table>

Two different transducers were used during the experiment. The micro-manometer can only read single pressures (or the difference between two) however it is more accurate than the transducer which was used in a Scanivalve system. The Scanivalve system is a multiplexing pressure sensor, which was adjusted to use an external transducer for the low pressure measurements. It has the ability to read up to 48 ports autonomously and record values as it goes. This transducer was used for the majority of the pressure measurements during the test. To reduce noise and small fluctuations, 50 measurements were taken at each port, and the average of those were used in the calculations. The micro-manometer was used for the individual pressure measurements, and for calibration of the Scanivalve transducer.

Table 6 - Uncertainty analysis

<table>
<thead>
<tr>
<th>0th Order</th>
<th>Uncertainty (+/-)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Pressure*</td>
<td>3.5x10^-3 in H2O (~0.6% of inlet dynamic head)</td>
</tr>
<tr>
<td>Temperature</td>
<td>1°C</td>
</tr>
<tr>
<td>Density</td>
<td>0.3%</td>
</tr>
<tr>
<td></td>
<td></td>
</tr>
<tr>
<td>--------------------------</td>
<td>-------</td>
</tr>
<tr>
<td>Flow Rate</td>
<td>1.1%</td>
</tr>
<tr>
<td>Reynolds Number</td>
<td>1.4%</td>
</tr>
<tr>
<td>Cp</td>
<td>2.7%</td>
</tr>
</tbody>
</table>

* - Pressure uncertainty was based on the Scanivalve transducer, the micro-manometer has a better accuracy if necessary.

For this experiment, the uncertainty values can be calculated based on the accuracy of the measurement devices. The uncertainty calculation is based on the following equation (Kline & McClintock, 1953):

$$
\Omega_y := \sum_{i=1}^{N} \frac{d}{dx_i} \left( y(x_1, x_2, ..., x_N) \Omega_{x,i} \right)^2
$$

(3)

Table 6 describes each measurement and the respective uncertainty.

**Experimental Validation**

To establish confidence in the experimental setup, a smooth pipe friction factor validation case was run. The diffuser and ECB was removed from the system, and replaced with a 60 diameter long smooth pipe instrumented with pressure taps along the wall. Operating at a Reynolds number of 200,000, the friction factor was obtained from (McKeon, Swanson, & Zagarola, 2004) giving a fully developed pressure drop as a function of pipe length. This result was plotted against the static pressure along the wall of the smooth pipe and is shown in Figure 33. The developing entrance length is easily distinguished from the data, and the experimental pressure in the fully developed region.
has a maximum deviation of 2.5% from the data obtained from (McKeon, Swanson, & Zagarola, 2004).

**Figure 33 -** Smooth pipe validation of the experimental setup. Static pressure normalized by the dynamic head (using bulk velocity, $V_b$) versus length normalized by the pipe diameter ($D_p$)

**Diffuser Cp Normalization**

The standard normalization of the pressure recovery coefficient, $C_p$, involves using the average static pressure at the inlet of the diffuser. Ideally, this value would be the area averaged static pressure of the entire flow area at the axial location of the beginning of the diffusion process. This value is simple to obtain from a computational source, however it is extremely difficult to measure this value in an experiment. A full
traverse of the inlet of the diffuser would be required for every operating condition of each case that is being investigated. The effort necessary to design a prove traverse system capable of traversing around the struts was outside the scope of the current experiment. Therefore, an alternate way of normalizing the Cp was developed.

Due to the manufacturing and assembly process of the inner diameter of the diffuser system, pressure taps were not an option for the ID surface. This only leaves the OD surface of the entrance section and the diffuser itself. From this, the most logical choice is to measure the average static pressure near the wall of the OD surface just prior to the inlet of the diffuser. While this may seem harmless, this can actually cause a significant discrepancy in the normalization process. Show in Figure 34 is a simple CFD case using a naturally separated (15° single wall expanding) diffuser. This case was chosen as it most accurately represents the region close to the inlet of the diffuser in the current study. What can be observed is that the flow near the expanding edge is locally accelerated around the corner, thus reducing its static pressure. While this seems counterintuitive, as diffusers are intended to increase pressure, the fluid particles near the wall have a longer axial distance to travel compared to the fluid particles further away from the wall. This distance is only a function of the turning angle of the diffuser wall, hence the larger the diffusion angle the larger the magnitude of the pressure drop locally at the edge of the inlet.

As an example on the magnitude of the erroneous Cp normalization, the case shown in Figure 34 was evaluated to determine the Cp of the diffuser using the standard normalization technique (area averaged inlet pressure) and the technique used in this study (local static pressure at the inlet OD) and is shown in Figure 35.
Immediately noticeable is the local measurement of the static pressure near the expanding wall (-166 Pa in this case) is lower than the area averaged pressure at the inlet (-107 Pa in this case). Therefore, due to the normalization procedure, the Cp values for the current study are larger than the actual Cp. This result should be noted throughout the report, as Cp values larger than 1 are reported (which is physically impossible without work input). While this normalization procedure does cause the magnitude of the Cp value to increase compared to its real value, the delta differences between each case still remain equivalent. Since the purpose of this study is to discover the differences between the cases chosen, both experimentally and computationally, this normalization procedure is adequate.
Figure 34 - Example of the effect of normalizing Cp using the expanding wall in a diffuser
Figure 35 - Cp Normalization comparison
CHAPTER 4: COMPUTATIONAL MODEL SETUP AND BOUNDARY CONDITIONS

As with any experiment, the obtainable data is limited by many factors. In this study, the aerodynamic characteristics were obtained using static pressure ports along the walls of the diffuser and of the ECB. In order to reduce the cost and complexity of the rig, pressure taps were not included on the inner diameter of the diffuser nor on the struts within it. Additionally, in order to keep the flow undisturbed within the diffuser (which is highly sensitive to disturbances), probes were not mounted within the flow field in the diffuser or in the ECB. This restriction severely limits the amount of data obtainable from the experimental rig. As a substitution to this, a computational model was created to try and predict the physics within the flow path. This CFD model will not only be used to fill in the gaps of the experiment, but it will also be used as a validation tool for the turbulence models and discretization schemes used in the computational code.

In the gas turbine industry, tools are designed and used in order to predict the performance of specific parts of the engine as accurately as possible. These predictions are used, in some cases, as a design tool in order to iteratively determine the best performing part which will eventually go into an engine if its capabilities are acceptable. Companies are currently using CFD as a design tool more and more often, as a high fidelity computational solution is becoming obtainable in a reasonable amount of time due to the increased computing performance with new technologies. In the past, these technologies were nearly as accurate as they are now, however the extensive amount
of time and effort necessary to obtain a solution for a single problem warranted them useless to a schedule driven industry. Now that a relatively inexpensive series of computers can predict a reasonably complex flow field within a matter of days, rather than weeks, it is possible to use as a design tool rather than an academic tool. However, there are two sides to this story. While CFD is becoming more and more accurate as time goes on, it still is not perfect and has its downfalls if used in the wrong way. As with an experiment, a poorly performed computational model can give erroneous results which could possibly drive the designer to an inferior solution to the problem at hand. Additionally, due to the complexity in the discretization schemes and turbulence models, CFD can be extremely accurate in a certain flow field while simultaneously poor in another. It is the job of the engineer to determine whether or not the tool being used is applicable and accurate enough to do the job properly before using it. This study aims to determine the accuracy of a very common computational model which is used widely throughout the gas turbine design industry: steady RANS with the k-ε turbulence model. This model is particularly good at capturing steady state flow fields, specifically axisymmetric jets. The model has been tuned by researchers over the past years to attempt to predict the spreading of a jet into quiescent air, and it has proven its accuracy in many flow fields. The downside to the k–ε turbulence model is its inaccuracy in predicting wall bounded flows, specifically separated diffusing flows. This turbulence model was chosen for this study, as it is a very widely used turbulence model, however not many understand the drawbacks to the near wall function approximations within the model. This study is here to show how well a typical industry
tool can predict a complex, separated, diffusing flow field and to determine whether or not the tool is worthy of such a use.

The main focus of this study is to investigate the accuracy of CFD calculations with today’s commercial CFD packages when applied to a diffuser exhaust system. The experimental tests were used as a baseline to compare against the data obtained from the computations. The inlet and exit boundary conditions were matched from the experimental conditions for each case. All data obtained from the computational domain was analyzed at the same physical locations on the diffuser and ECB as the experimental rig. The commercial software StarCCM+ v5.04 (Star-CCM, 2010) was used for the mesh generation and the segregated, steady, incompressible Navier-Stokes solver for this investigation.

An all wall Y+ treatment was used in this model to allow the use of wall cell heights in the diffuser and ECB in the viscous sub-layer (Y+ < 5) and wall cell heights in the exit section in the fully turbulent region (Y+ > 30). The code uses the appropriate wall function depending on the value of Y+. Cells in the intermediate region between these zones are interpolated; however there were only a few cells (less than 0.001% of total cell count) in this intermediate Y+ region within the diffuser and ECB.

The realizable k-ε model was used in this study with all model constants set as the default values in StarCCM. This model was also used in [ (Sultanian, Nagao, & Sakamoto, Experimental and Three-Dimensional CFD Investigation in a Gas Turbine Exhaust System, 1999), (Sultanian & Mongia, Fuel Nozzle Air Flow Modeling, 1986), (Hirsch & Khodak, 1995)] under very similar adverse pressure gradient flow fields with successful results. The convergence criteria for the computational model were set at a
minimum residual of $10^{-6}$ for all equations. Iterative convergence was also monitored for the average total pressure at the exit of the diffuser, obtaining convergence when the total pressure change between iterations was less than $10^{-3}$. The model convergence was verified when both of these conditions were satisfied. The convergence criteria require approximately 2000-3000 iterations per case.

**Inlet Condition**

It is no secret that diffusers in general are highly sensitive to the inlet condition. This is no different for a computational model than it is for an experiment. Since a full inlet profile was not available for each case studied, two options were available for the inlet boundary condition of the computational model. The first option is modeling far enough upstream of the diffuser to accurately capture the physics within the plenum and nozzle assembly, thus create its own inlet profile at the inlet of the diffuser. This option is by far the most robust way of calculating the inlet condition of a diffuser with varying geometries, as the computation is allowed to predict changes in the inlet condition due to geometric variations within the diffuser and downstream of it. Unfortunately, in order to accurately model the inlet flow system a excessive amount of volume must be added to the system. The flow volume that is being calculated would need to be nearly double of what it is for the diffuser and ECB if part the inlet system were to be included. This would increase the number of cells, thus the number of equations and unknowns that need to be solved for every iteration of every case being run. The increased time and computational resources required were not feasible for this study; therefore a second option for applying boundary conditions was derived. While it was out of the scope of
this study to create a full inlet model for every case, it was not out of the question to create and solve this model once.

A CFD model was created of the entire experimental rig, from plenum inlet to ECB exhaust, in order to determine the appropriate boundary condition for the diffuser inlet. The model was created using a baseline geometry of the diffuser and ECB, which accurately described the exit boundary conditions of the diffusing flow field. This was chosen in order to have a representative inlet condition under the assumption that this condition would not change significantly when the exit boundary condition changed with varying ECB geometries. The mesh was created using the same conditions as the mesh used in each case in the study, in order to alleviate any mesh interpolation errors caused by mapping data onto different sized cells. The model used is shown in Figure 36.

Figure 36 - Full experiment model used for the CFD inlet boundary condition
The entire model consisted of over 6 million cells, nearly half of which was in the flow conditioning and exhaust sections of the model. The boundary conditions used for this model were easily obtainable from the experiment. At the inlet of the plenum upstream of the diffuser, a mass flow of 0.25 kg/s was applied as this is the target mass flow for each case in this study. The only other boundary condition was at the exit of the ECB, where the dump plenum was set to have a constant boundary static pressure set to atmospheric, as the experimental rig dumps the flow into the atmosphere after the ECB. All other boundaries within the model were set as walls with the standard no-slip condition. The benefit of having such a large model, is the ease of boundary condition definition, however the increased computation time overpowers this convenience most of the time. This model was then initialized and solved over approximately 1 week using a single computer and the boundary condition at the inlet of the diffuser was extracted, and is shown in Figure 37.
Interestingly, even though all geometry upstream of the diffuser is geometrically axisymmetric, the inlet boundary condition of the diffuser is not. It is observed that the 12:00 location of the inlet has a higher velocity than the 6:00 location. This is due to the asymmetric exit boundary condition of the diffuser. Even through a separated diffusing flow field, the exit boundary condition is imposing an asymmetric boundary condition at the inlet of the diffuser. The high pressure zone at the exit of the diffuser at the 6:00 location as it dumps into the ECB restricts the flow rate though this section of the diffuser. Since the fluid prefers the path of least resistance, more fluid is pushed

Figure 37 - CFD inlet boundary condition for the diffuser
towards the 12:00 location, thus promoting a lower pressure and a higher velocity through the 12:00 location compared to the 6:00 location (see Figure 38).

![Figure 38 - Pressure contours (left) and Velocity vectors (right) for the inlet conditioning CFD model](image)

This inlet condition was extracted from the full model, and applied as the inlet condition to each case modeled for the remainder of the computational study. The assumption made here, is that this inlet condition remains constant throughout the geometric changes happening downstream of the diffuser during the course of the study. The accuracy of this assumption will be evaluated once the study has been completed.

**Mesh and Boundary Conditions**  
Once the inlet condition was obtained, the remaining boundary conditions could be defined for the CFD model. It was noticed that each geometry chosen for this study, a
geometric symmetry plane was available at the 6:00 to 12:00 plane. An assumption was made based on previous studies done (Bernier, Riclick, & Kapat, 2011) that the flow field through the diffuser and ECB system was geometrically similar about this symmetry plane. Therefore, the entire model was cut in half and a symmetry boundary condition was applied at this location. This boundary condition lets no mass pass through the boundary, and the derivative and second derivative of all variables are set to zero at this plane. Physically, this means that there are no discontinuous jumps in pressure or velocity (or any other variable) across the symmetry plane, and all solutions can simply be mirrored about this plane. Again, this is an assumption made during the modeling process that enables a more dense mesh in the computational volume due to keeping the same total number of cells, but only using half the volume. This enables a more detailed solution in the fluid volume without increasing the time required to solve the equations. The remaining boundary condition was the atmospheric pressure dump at the exit of the ECB, which remained unchanged from the full model. The finalized model and boundary conditions are shown in Figure 39
A grid dependence study was used in this study to determine the effect of the mesh size on the solution. A perfect grid study would show the point at which an increased number of cells no longer changes the solution. This point would then be used as a mesh size indicator for the remaining CFD models in the study, removing the variable of a change in the solution due to a change in the mesh, hence the term grid convergence. In order to evaluate this converged state, a number of models must be created and solved with varying grid sizes. For this study, 6 different mesh sizes were evaluated for a single geometry with equivalent boundary conditions and equations for each case. The total
number of cells ranged through an entire order of magnitude, from just over 500,000 to 5,000,000 cells. Models with more cells were not evaluated, as the estimated time to complete the required number of cases at the increased cell count was determined to be out of the scope of this study. The variables which were chosen to be compared between each of the cases was the system performance values that are the goal of the study as a whole, being the system total pressure loss and the diffuser pressure recovery. The results of the grid dependence study are shown in Figure 40.
Immediately noticeable is that for both the diffuser pressure recovery and the system total pressure loss, the grid is not converged. In fact, the solutions are asymptotically approaching the converged value at an extremely slow rate. Even if the cell count was increased by a factor of 2 to nearly 10,000,000 it still may not be completely converged. Unfortunately, the study must continue even with this result. It should be noted for the remainder of this study that the grid is not completely converged, however an increase in cell count of nearly 2 (from 2,750,000 to 5,000,000) accounts for approximately 1% decrease in the diffuser pressure recovery and a 1% increase in the system total pressure loss. From this result alone, it can be theorized that the physics being modeled within the diffuser are not complete. The smaller the cell size becomes, the more eddies that are resolved which account for a higher mixing loss through the separated zones of the diffuser, thus decreasing the pressure recovery possible in the diffuser and increasing the total pressure lost through the system. It is theorized that an increase in the number of cells, as well as an increase in the fidelity of the turbulence modeling (i.e. changing from a steady RANS, to an unsteady RANS, to a large eddy simulation, to direct numerical simulation) will continuously increase the loss through a diffusing system, reducing the static pressure recovery and increasing the total pressure lost in the system. For this study, accounting for the fact that some cases have a volume that is nearly 50% larger than this grid dependence model, the nominal grid size was chosen at the 2.75 million cell mark. This cell size corresponded to wall y+ values less than 6 for the entire volume of the test section, only increasing on the exit boundary faces outside of the ECB. For the majority of the diffuser, the wall y+ values were less than 1 with the exception to about 10-20 cells at the stagnation point of the struts, due to the stagnation
point anomaly when defining the u+ value. On a case to case basis, the nominal cell volume was kept constant as well as the surface size of the elements on each wall of the test section. The variation in the number of cells is only due to the variation in volume from one case to another. The total cell count for the cases ranged from a minimum of 2.1 million (Case 9), to a maximum of 3.9 million (Case 11). A few images of the final mesh for a typical case are shown in Figure 41, Figure 42 and Figure 43.
Figure 42 - CFD model mesh looking at the struts of the diffuser from within the air volume

Figure 43 - CFD model mesh of the diffuser inlet
CHAPTER 3: RESULTS

Diffuser with no ECB

Diffuser Inlet Profiles (Free Exhaust)

Figure 44 - Diffuser inlet profiles for the diffuser with no ECB (Free Exhaust)

Figure 45 - Pressure recovery along the outer annulus of the diffuser with no ECB attached
Figure 46 - Experimental and computational results for the velocity profile with no ECB from the ID \([(r-Ri)/(Ro-Ri)=0]\) to OD \([(r-Ri)/(Ro-Ri)=1]\) at the exit between the struts (top), at the exit behind the struts (mid) and at the diffuser inlet (bottom). Velocity is on the left axis, and turbulence intensity ratio is on the right axis.
Table 7 - Local average turbulence intensity in the diffuser

<table>
<thead>
<tr>
<th>Location</th>
<th>Experiment</th>
<th>Computation</th>
</tr>
</thead>
<tbody>
<tr>
<td>Inlet</td>
<td>1.1</td>
<td>1.3</td>
</tr>
<tr>
<td>Exit (Behind Strut)</td>
<td>33.1</td>
<td>17.8</td>
</tr>
<tr>
<td>Exit (Between Strut)</td>
<td>26.5</td>
<td>14.5</td>
</tr>
</tbody>
</table>

Figure 47 - Axial velocity profile at the exit of the diffuser under a free exit condition showing the asymmetric separation zones
Experiment - Case 1

**Figure 48** - Circumferential static pressure at the inlet of the diffuser for the experimental Case 1

**Figure 49** - Diffuser pressure recovery on the OD for the experimental Case 1
Figure 50 - ECB wall pressures on the diffuser's side for the experimental Case 1

Figure 51 - ECB wall pressures on the impingement side for the experimental Case 1
Figure 52 - Circumferential static pressure at the inlet of the diffuser for the experimental Case 2

Figure 53 - Diffuser pressure recovery on the OD for the experimental Case 2
Figure 54 - ECB wall pressures on the diffuser's side for the experimental Case 2

Figure 55 - ECB wall pressures on the impingement side for the experimental Case 2
Figure 56 - Circumferential static pressure at the inlet of the diffuser for the experimental Case 3

Figure 57 - Diffuser pressure recovery on the OD for the experimental Case 3
**Figure 58** - ECB wall pressures on the diffuser's side for the experimental Case 3

**Figure 59** - ECB wall pressures on the impingement side for the experimental Case 3
Diffuser Inlet OD Static Pressure

Figure 60 - Circumferential static pressure at the inlet of the diffuser for the experimental Case 4

Diffuser Pressure Recovery

Figure 61 - Diffuser pressure recovery on the OD for the experimental Case 4
Figure 62 - ECB wall pressures on the diffuser's side for the experimental Case 4

Figure 63 - ECB wall pressures on the impingement side for the experimental Case 4
Figure 64 - Circumferential static pressure at the inlet of the diffuser for the experimental Case 5

Figure 65 - Diffuser pressure recovery on the OD for the experimental Case 5
Figure 66 - ECB wall pressures on the diffuser's side for the experimental Case 5

Figure 67 - ECB wall pressures on the impingement side for the experimental Case 5
Experiment - Case 6

**Diffuser Inlet OD Static Pressure**

Figure 68 - Circumferential static pressure at the inlet of the diffuser for the experimental Case 6

**Diffuser Pressure Recovery**

Figure 69 - Diffuser pressure recovery on the OD for the experimental Case 6
Figure 70 - ECB wall pressures on the diffuser's side for the experimental Case 6

Figure 71 - ECB wall pressures on the impingement side for the experimental Case 6
Experiment - Case 7

**Diffuser Inlet OD Static Pressure**

Figure 72 - Circumferential static pressure at the inlet of the diffuser for the experimental Case 7

**Diffuser Pressure Recovery**

Figure 73 - Diffuser pressure recovery on the OD for the experimental Case 7
Figure 74 - ECB wall pressures on the diffuser's side for the experimental Case 7

Figure 75 - ECB wall pressures on the impingement side for the experimental Case 7
Figure 76 - Circumferential static pressure at the inlet of the diffuser for the experimental Case 8

Figure 77 - Diffuser pressure recovery on the OD for the experimental Case 8
Figure 78 - ECB wall pressures on the diffuser's side for the experimental Case 8

Figure 79 - ECB wall pressures on the impingement side for the experimental Case 8
Experiment - Case 9

Diffuser Inlet OD Static Pressure

Figure 80 - Circumferential static pressure at the inlet of the diffuser for the experimental Case 9

Diffuser Pressure Recovery

Figure 81 - Diffuser pressure recovery on the OD for the experimental Case 9
Figure 82 - ECB wall pressures on the diffuser's side for the experimental Case 9

Figure 83 - ECB wall pressures on the impingement side for the experimental Case 9
Experiment - Case 10

Diffuser Inlet OD Static Pressure

Figure 84 - Circumferential static pressure at the inlet of the diffuser for the experimental Case 10

Diffuser Pressure Recovery

Figure 85 - Diffuser pressure recovery on the OD for the experimental Case 10
Figure 86 - ECB wall pressures on the diffuser's side for the experimental Case 10

Figure 87 - ECB wall pressures on the impingement side for the experimental Case 10
Experiment - Case 11

**Figure 88** - Circumferential static pressure at the inlet of the diffuser for the experimental Case 11

**Figure 89** - Diffuser pressure recovery on the OD for the experimental Case 11
Figure 90 - ECB wall pressures on the diffuser's side for the experimental Case 11

Figure 91 - ECB wall pressures on the impingement side for the experimental Case 11
Figure 92 - Circumferential static pressure at the inlet of the diffuser for the experimental Case 12

Figure 93 - Diffuser pressure recovery on the OD for the experimental Case 12
Figure 94 - ECB wall pressures on the diffuser's side for the experimental Case 12

Figure 95 - ECB wall pressures on the impingement side for the experimental Case 12
Experiment - Case 13

Diffuser Inlet OD Static Pressure

Figure 96 - Circumferential static pressure at the inlet of the diffuser for the experimental Case 13

Diffuser Pressure Recovery

Figure 97 - Diffuser pressure recovery on the OD for the experimental Case 13
Figure 98 - ECB wall pressures on the diffuser's side for the experimental Case 13

Figure 99 - ECB wall pressures on the impingement side for the experimental Case 13
Figure 100 - Case 1 Geometry

Figure 101 - Diffuser pressure recovery on the OD for the CFD Case 1
Figure 102 - ECB wall pressures on the diffuser's side for the CFD Case 1

Figure 103 - ECB wall pressures on the impingement side for the CFD Case 1
Figure 104 - Symmetry plane velocity vectors for the CFD Case 1
Figure 105 - Diffuser inlet and exit velocity vectors for the CFD Case 1

Figure 106 - Breakout plane velocity vectors for the CFD Case 1
Figure 107 - ECB exit velocity vectors for the CFD Case 1

Figure 108 - Diffuser inlet and exit pressure profiles for the CFD Case 1
Figure 109 - Diffuser OD wall pressure for the CFD Case 1

Figure 110 - ECB wall pressure on the diffuser side for the CFD Case 1
Figure 111 - ECB wall pressure for the impingement side for the CFD Case 1

Figure 112 - ECB exit total pressure profile for the CFD Case 1
Figure 113 - Symmetry plane pressure profile for the CFD Case 1
Figure 114 - Case 2 Geometry

Figure 115 - Diffuser pressure recovery on the OD for the CFD Case 2
Figure 116 - ECB wall pressures on the diffuser's side for the CFD Case 2

Figure 117 - ECB wall pressures on the impingement side for the CFD Case 2
Figure 118 - Symmetry plane velocity vectors for the CFD Case 2
Figure 119 - Diffuser inlet and exit velocity vectors for the CFD Case 2

Figure 120 - Breakout plane velocity vectors for the CFD Case 2
Figure 121 - ECB exit velocity vectors for the CFD Case 2

Figure 122 - Diffuser inlet and exit pressure profiles for the CFD Case 2
Figure 123 - Diffuser OD wall pressure for the CFD Case 2

Figure 124 - ECB wall pressure on the diffuser side for the CFD Case 2
Figure 125 - ECB wall pressure for the impingement side for the CFD Case 2

Figure 126 - ECB exit total pressure profile for the CFD Case 2
Figure 127 - Symmetry plane pressure profile for the CFD Case 2
Figure 128 - Case 3 Geometry

Figure 129 - Diffuser pressure recovery on the OD for the CFD Case 3
Figure 130 - ECB wall pressures on the diffuser's side for the CFD Case 3

Figure 131 - ECB wall pressures on the impingement side for the CFD Case 3
Figure 132 - Symmetry plane velocity vectors for the CFD Case 3
Figure 133 - Diffuser inlet and exit velocity vectors for the CFD Case 3

Figure 134 - Breakout plane velocity vectors for the CFD Case 3
Figure 135 - ECB exit velocity vectors for the CFD Case 3

Figure 136 - Diffuser inlet and exit pressure profiles for the CFD Case 3
Figure 137 - Diffuser OD wall pressure for the CFD Case 3

Figure 138 - ECB wall pressure on the diffuser side for the CFD Case 3
Figure 139 - ECB wall pressure for the impingement side for the CFD Case 3

Figure 140 - ECB exit total pressure profile for the CFD Case 3
Figure 141 - Symmetry plane pressure profile for the CFD Case 3
Figure 142 - Case 4 Geometry

Figure 143 - Diffuser pressure recovery on the OD for the CFD Case 4
Figure 144 - ECB wall pressures on the diffuser's side for the CFD Case 4

Figure 145 - ECB wall pressures on the impingement side for the CFD Case 4
Figure 146 - Symmetry plane velocity vectors for the CFD Case 4
Figure 147 - Diffuser inlet and exit velocity vectors for the CFD Case 4

Figure 148 - Breakout plane velocity vectors for the CFD Case 4
Figure 149 - ECB exit velocity vectors for the CFD Case 4

Figure 150 - Diffuser inlet and exit pressure profiles for the CFD Case 4
Figure 151 - Diffuser OD wall pressure for the CFD Case 4

Figure 152 - ECB wall pressure on the diffuser side for the CFD Case 4
Figure 153 - ECB wall pressure for the impingement side for the CFD Case 4

Figure 154 - ECB exit total pressure profile for the CFD Case 4
Figure 155 - Symmetry plane pressure profile for the CFD Case 4
Figure 156 - Case 5 Geometry

Figure 157 - Diffuser pressure recovery on the OD for the CFD Case 5
Figure 158 - ECB wall pressures on the diffuser's side for the CFD Case 5

Figure 159 - ECB wall pressures on the impingement side for the CFD Case 5
Figure 160 - Symmetry plane velocity vectors for the CFD Case 5
Figure 161 - Diffuser inlet and exit velocity vectors for the CFD Case 5

Figure 162 - Breakout plane velocity vectors for the CFD Case 5
Figure 163 - ECB exit velocity vectors for the CFD Case 5

Figure 164 - Diffuser inlet and exit pressure profiles for the CFD Case 5
Figure 165 - Diffuser OD wall pressure for the CFD Case 5

Figure 166 - ECB wall pressure on the diffuser side for the CFD Case 5
Figure 167 - ECB wall pressure for the impingement side for the CFD Case 5

Figure 168 - ECB exit total pressure profile for the CFD Case 5
Figure 169 - Symmetry plane pressure profile for the CFD Case 5
Figure 170 - Case 6 Geometry

Figure 171 - Diffuser pressure recovery on the OD for the CFD Case 6
Figure 172 - ECB wall pressures on the diffuser's side for the CFD Case 6

Figure 173 - ECB wall pressures on the impingement side for the CFD Case 6
Figure 174 - Symmetry plane velocity vectors for the CFD Case 6
Figure 175 - Diffuser inlet and exit velocity vectors for the CFD Case 6

Figure 176 - Breakout plane velocity vectors for the CFD Case 6
Figure 177 - ECB exit velocity vectors for the CFD Case 6

Figure 178 - Diffuser inlet and exit pressure profiles for the CFD Case 6
Figure 179 - Diffuser OD wall pressure for the CFD Case 6

Figure 180 - ECB wall pressure on the diffuser side for the CFD Case 6
Figure 181 - ECB wall pressure for the impingement side for the CFD Case 6

Figure 182 - ECB exit total pressure profile for the CFD Case 6
Figure 183 - Symmetry plane pressure profile for the CFD Case 6
Case 7

Figure 184 - Case 7 Geometry

Diffuser Pressure Recovery

Figure 185 - Diffuser pressure recovery on the OD for the CFD Case 7
Figure 186 - ECB wall pressures on the diffuser's side for the CFD Case 7

Figure 187 - ECB wall pressures on the impingement side for the CFD Case 7
Figure 188 - Symmetry plane velocity vectors for the CFD Case 7
Figure 189 - Diffuser inlet and exit velocity vectors for the CFD Case 7

Figure 190 - Breakout plane velocity vectors for the CFD Case 7
Figure 191 - ECB exit velocity vectors for the CFD Case 7

Figure 192 - Diffuser inlet and exit pressure profiles for the CFD Case 7
Figure 193 - Diffuser OD wall pressure for the CFD Case 7

Figure 194 - ECB wall pressure on the diffuser side for the CFD Case 7
Figure 195 - ECB wall pressure for the impingement side for the CFD Case 7

Figure 196 - ECB exit total pressure profile for the CFD Case 7
Figure 197 - Symmetry plane pressure profile for the CFD Case 7
**CFD – Case 8**

Figure 198 - Case 8 Geometry

**Diffuser Pressure Recovery**

Figure 199 - Diffuser pressure recovery on the OD for the CFD Case 8
Figure 200 - ECB wall pressures on the diffuser’s side for the CFD Case 8

Figure 201 - ECB wall pressures on the impingement side for the CFD Case 8
Figure 202 - Symmetry plane velocity vectors for the CFD Case 8
Figure 203 - Diffuser inlet and exit velocity vectors for the CFD Case 8

Figure 204 - Breakout plane velocity vectors for the CFD Case 8
Figure 205 - ECB exit velocity vectors for the CFD Case 8

Figure 206 - Diffuser inlet and exit pressure profiles for the CFD Case 8
Figure 207 - Diffuser OD wall pressure for the CFD Case 8

Figure 208 - ECB wall pressure on the diffuser side for the CFD Case 8
Figure 209 - Rescaled image of the ECB wall pressure on the diffuser side for the CFD Case 8

Figure 210 - ECB wall pressure for the impingement side for the CFD Case 8
Figure 211 - Rescaled image of the ECB wall pressure for the impingement side for the CFD Case 8

Figure 212 - ECB exit total pressure profile for the CFD Case 8
Figure 213 - Symmetry plane pressure profile for the CFD Case 8
Figure 214 - Case 9 Geometry

Figure 215 - Diffuser pressure recovery on the OD for the CFD Case 9
Figure 216 - ECB wall pressures on the diffuser's side for the CFD Case 9

Figure 217 - ECB wall pressures on the impingement side for the CFD Case 9
Figure 218 - Symmetry plane velocity vectors for the CFD Case 9
Figure 219 - Diffuser inlet and exit velocity vectors for the CFD Case 9

Figure 220 - Breakout plane velocity vectors for the CFD Case 9
Figure 221 - ECB exit velocity vectors for the CFD Case 9

Figure 222 - Diffuser inlet and exit pressure profiles for the CFD Case 9
Figure 223 - Diffuser OD wall pressure for the CFD Case 9

Figure 224 - ECB wall pressure on the diffuser side for the CFD Case 9
Figure 225 - Rescaled image of the ECB wall pressure on the diffuser side for the CFD Case 9

Figure 226 - ECB wall pressure for the impingement side for the CFD Case 9
Figure 227 - Rescaled image of the ECB wall pressure for the impingement side for the CFD Case 9

Figure 228 - ECB exit total pressure profile for the CFD Case 9
Figure 229 - Symmetry plane pressure profile for the CFD Case 9
Figure 230 - Case 10 Geometry

Figure 231 - Diffuser pressure recovery on the OD for the CFD Case 10
Figure 232 - ECB wall pressures on the diffuser's side for the CFD Case 10

Figure 233 - ECB wall pressures on the impingement side for the CFD Case 10
Figure 234 - Symmetry plane velocity vectors for the CFD Case 10
Figure 235 - Diffuser inlet and exit velocity vectors for the CFD Case 10

Figure 236 - Breakout plane velocity vectors for the CFD Case 10
Figure 237 - ECB exit velocity vectors for the CFD Case 10

Figure 238 - Diffuser inlet and exit pressure profiles for the CFD Case 10
Figure 239 - Diffuser OD wall pressure for the CFD Case 10

Figure 240 - ECB wall pressure on the diffuser side for the CFD Case 10
Figure 241 - ECB wall pressure for the impingement side for the CFD Case 10

Figure 242 - ECB exit total pressure profile for the CFD Case 10
Figure 243 - Symmetry plane pressure profile for the CFD Case 10
Figure 244 - Case 11 Geometry

Figure 245 - Diffuser pressure recovery on the OD for the CFD Case 11
Figure 246 - ECB wall pressures on the diffuser's side for the CFD Case 11

Figure 247 - ECB wall pressures on the impingement side for the CFD Case 11
Figure 248 - Symmetry plane velocity vectors for the CFD Case 11
Figure 249 - Diffuser inlet and exit velocity vectors for the CFD Case 11

Figure 250 - Breakout plane velocity vectors for the CFD Case 11
Figure 251 - ECB exit velocity vectors for the CFD Case 11

Figure 252 - Diffuser inlet and exit pressure profiles for the CFD Case 11
Figure 253 - Diffuser OD wall pressure for the CFD Case 11

Figure 254 - ECB wall pressure on the diffuser side for the CFD Case 11
Figure 255 - ECB wall pressure for the impingement side for the CFD Case 11

Figure 256 - ECB exit total pressure profile for the CFD Case 11
Figure 257 - Symmetry plane pressure profile for the CFD Case 11
Figure 258 - Case 12 Geometry

Figure 259 - Diffuser pressure recovery on the OD for the CFD Case 12
Figure 260 - ECB wall pressures on the diffuser's side for the CFD Case 12.

Figure 261 - ECB wall pressures on the impingement side for the CFD Case 12.
Figure 262 - Symmetry plane velocity vectors for the CFD Case 12
Figure 263 - Diffuser inlet and exit velocity vectors for the CFD Case 12

Figure 264 - Breakout plane velocity vectors for the CFD Case 12
Figure 265 - ECB exit velocity vectors for the CFD Case 12

Figure 266 - Diffuser inlet and exit pressure profiles for the CFD Case 12
Figure 267 - Diffuser OD wall pressure for the CFD Case 12

Figure 268 - ECB wall pressure on the diffuser side for the CFD Case 12
Figure 269 - ECB wall pressure for the impingement side for the CFD Case 12

Figure 270 - ECB exit total pressure profile for the CFD Case 12
Figure 271 - Symmetry plane pressure profile for the CFD Case 12
Figure 272 - Case 13 Geometry

Figure 273 - Diffuser pressure recovery on the OD for the CFD Case 13
Figure 274 - ECB wall pressures on the diffuser's side for the CFD Case 13

Figure 275 - ECB wall pressures on the impingement side for the CFD Case 13
Figure 276 - Symmetry plane velocity vectors for the CFD Case 13
Figure 277 - Diffuser inlet and exit velocity vectors for the CFD Case 13

Figure 278 - Breakout plane velocity vectors for the CFD Case 13
Figure 279 - ECB exit velocity vectors for the CFD Case 13

Figure 280 - Diffuser inlet and exit pressure profiles for the CFD Case 13
Figure 281 - Diffuser OD wall pressure for the CFD Case 13

Figure 282 - ECB wall pressure on the diffuser side for the CFD Case 13
Figure 283 - ECB wall pressure for the impingement side for the CFD Case 13

Figure 284 - ECB exit total pressure profile for the CFD Case 13
Figure 285 - Symmetry plane pressure profile for the CFD Case 13
Box Behnken Design Results

Experimental Results

Contour Plot of Inlet Pressure Variation vs %ECB Width, %ECB Length

Figure 286 - Experimental Box Behnken Response Surface for Inlet Pressure Variation
Figure 287 - Experimental Box Behnken Response Surface for Inlet Pressure Variation

Contour Plot of Inlet Pressure Variation vs %Protrusion, %ECB Length

Figure 288 - Experimental Box Behnken Response Surface for Inlet Pressure Variation

Contour Plot of Inlet Pressure Variation vs %Protrusion, %ECB Width
Figure 289 - Experimental Box Behnken Response Surface for System Cp

Figure 290 - Experimental Box Behnken Response Surface for System Cp
Figure 291 - Experimental Box Behnken Response Surface for System Cp

Contour Plot of System Cp vs %Protrusion, %ECB Length

Figure 292 - Experimental Box Behnken Response Surface for System Total Pressure Loss

Contour Plot of System Pt Loss vs %ECB Width, %ECB Length

208
Figure 293 - Experimental Box Behnken Response Surface for System Total Pressure Loss

Figure 294 - Experimental Box Behnken Response Surface for System Total Pressure Loss
### Table 8 – Experimental Inlet Pressure Variation Coefficients

<table>
<thead>
<tr>
<th>Term</th>
<th>Coefficients</th>
<th>Contribution [%]</th>
</tr>
</thead>
<tbody>
<tr>
<td>Constant</td>
<td>$C_0$</td>
<td>0.173541</td>
</tr>
<tr>
<td>%ECB Length</td>
<td>$C_1$</td>
<td>-0.00271</td>
</tr>
<tr>
<td>%ECB Width</td>
<td>$C_2$</td>
<td>-0.03434</td>
</tr>
<tr>
<td>%Protrusion</td>
<td>$C_3$</td>
<td>-0.01419</td>
</tr>
<tr>
<td>%ECB Length * %ECB Length</td>
<td>$C_{11}$</td>
<td>-0.01828</td>
</tr>
<tr>
<td>%ECB Width * %ECB Width</td>
<td>$C_{22}$</td>
<td>0.003658</td>
</tr>
<tr>
<td>%Protrusion * %Protrusion</td>
<td>$C_{33}$</td>
<td>-0.01827</td>
</tr>
<tr>
<td>%ECB Length * %ECB Width</td>
<td>$C_{12}$</td>
<td>0.009231</td>
</tr>
<tr>
<td>%ECB Length * %Protrusion</td>
<td>$C_{13}$</td>
<td>-0.00797</td>
</tr>
<tr>
<td>%ECB Width * %Protrusion</td>
<td>$C_{23}$</td>
<td>0.055093</td>
</tr>
<tr>
<td><strong>Summation</strong></td>
<td></td>
<td><strong>100.0%</strong></td>
</tr>
</tbody>
</table>
### Table 9 - Experimental System Cp Coefficients

<table>
<thead>
<tr>
<th>Term</th>
<th>Coefficients</th>
<th>Contribution [%]</th>
</tr>
</thead>
<tbody>
<tr>
<td>Constant</td>
<td>$C_0$</td>
<td>0.67857</td>
</tr>
<tr>
<td>%ECB Length</td>
<td>$C_1$</td>
<td>0.48238</td>
</tr>
<tr>
<td>%ECB Width</td>
<td>$C_2$</td>
<td>0.14258</td>
</tr>
<tr>
<td>%Protrusion</td>
<td>$C_3$</td>
<td>-0.27837</td>
</tr>
<tr>
<td>%ECB Length*%ECB Length</td>
<td>$C_{11}$</td>
<td>-0.39279</td>
</tr>
<tr>
<td>%ECB Width*%ECB Width</td>
<td>$C_{22}$</td>
<td>0.02762</td>
</tr>
<tr>
<td>%Protrusion*%Protrusion</td>
<td>$C_{33}$</td>
<td>-0.1381</td>
</tr>
<tr>
<td>%ECB Length*%ECB Width</td>
<td>$C_{12}$</td>
<td>-0.13279</td>
</tr>
<tr>
<td>%ECB Length*%Protrusion</td>
<td>$C_{13}$</td>
<td>0.37953</td>
</tr>
<tr>
<td>%ECB Width*%Protrusion</td>
<td>$C_{23}$</td>
<td>0.03786</td>
</tr>
<tr>
<td>Summation</td>
<td></td>
<td><strong>100.0%</strong></td>
</tr>
</tbody>
</table>
Table 10 - Experimental System Total Pressure Loss Coefficients

<table>
<thead>
<tr>
<th>Term</th>
<th>Coefficients</th>
<th>Contribution [%]</th>
</tr>
</thead>
<tbody>
<tr>
<td>Constant</td>
<td>$C_0$</td>
<td>0.56479</td>
</tr>
<tr>
<td>%ECB Length</td>
<td>$C_1$</td>
<td>-0.47237</td>
</tr>
<tr>
<td>%ECB Width</td>
<td>$C_2$</td>
<td>-0.12689</td>
</tr>
<tr>
<td>%Protrusion</td>
<td>$C_3$</td>
<td>0.2809</td>
</tr>
<tr>
<td>%ECB Length * %ECB Length</td>
<td>$C_{11}$</td>
<td>0.37035</td>
</tr>
<tr>
<td>%ECB Width * %ECB Width</td>
<td>$C_{22}$</td>
<td>-0.02681</td>
</tr>
<tr>
<td>%Protrusion * %Protrusion</td>
<td>$C_{33}$</td>
<td>0.11669</td>
</tr>
<tr>
<td>%ECB Length * %ECB Width</td>
<td>$C_{12}$</td>
<td>0.12943</td>
</tr>
<tr>
<td>%ECB Length * %Protrusion</td>
<td>$C_{13}$</td>
<td>-0.37671</td>
</tr>
<tr>
<td>%ECB Width * %Protrusion</td>
<td>$C_{23}$</td>
<td>-0.03113</td>
</tr>
<tr>
<td><strong>Summation</strong></td>
<td></td>
<td><strong>100.0%</strong></td>
</tr>
</tbody>
</table>
Computational Results

Figure 295 - Computational Box Behnken Response Surface for System Cp

Figure 296 - Computational Box Behnken Response Surface for System Cp
Figure 297 - Computational Box Behnken Response Surface for System Cp

Figure 298 - Computational Box Behnken Response Surface for System Total Pressure Loss

214
Figure 299 - Computational Box Behnken Response Surface for System Total Pressure Loss

Figure 300 - Computational Box Behnken Response Surface for System Total Pressure Loss
<table>
<thead>
<tr>
<th>Term</th>
<th>Coefficients</th>
<th>Contribution [%]</th>
</tr>
</thead>
<tbody>
<tr>
<td>Constant</td>
<td>$C_0$</td>
<td>0.4726</td>
</tr>
<tr>
<td>%ECB Length</td>
<td>$C_1$</td>
<td>0.3394</td>
</tr>
<tr>
<td>%ECB Width</td>
<td>$C_2$</td>
<td>0.1230</td>
</tr>
<tr>
<td>%Protrusion</td>
<td>$C_3$</td>
<td>-0.2116</td>
</tr>
<tr>
<td>%ECB Length*%ECB Length</td>
<td>$C_{11}$</td>
<td>-0.2501</td>
</tr>
<tr>
<td>%ECB Width*%ECB Width</td>
<td>$C_{22}$</td>
<td>-0.0101</td>
</tr>
<tr>
<td>%Protrusion*%Protrusion</td>
<td>$C_{33}$</td>
<td>-0.1186</td>
</tr>
<tr>
<td>%ECB Length*%ECB Width</td>
<td>$C_{12}$</td>
<td>-0.1461</td>
</tr>
<tr>
<td>%ECB Length*%Protrusion</td>
<td>$C_{13}$</td>
<td>0.2546</td>
</tr>
<tr>
<td>%ECB Width*%Protrusion</td>
<td>$C_{23}$</td>
<td>0.0574</td>
</tr>
<tr>
<td>Summation</td>
<td></td>
<td>100.0%</td>
</tr>
<tr>
<td>Term</td>
<td>Coefficients</td>
<td>Contribution [%]</td>
</tr>
<tr>
<td>-----------------------------</td>
<td>--------------</td>
<td>------------------</td>
</tr>
<tr>
<td>Constant</td>
<td>$C_0$</td>
<td>0.4404</td>
</tr>
<tr>
<td>%ECB Length</td>
<td>$C_1$</td>
<td>-0.1834</td>
</tr>
<tr>
<td>%ECB Width</td>
<td>$C_2$</td>
<td>-0.0597</td>
</tr>
<tr>
<td>%Protrusion</td>
<td>$C_3$</td>
<td>0.1812</td>
</tr>
<tr>
<td>%ECB Length*%ECB Length</td>
<td>$C_{11}$</td>
<td>0.1289</td>
</tr>
<tr>
<td>%ECB Width*%ECB Width</td>
<td>$C_{22}$</td>
<td>-0.0013</td>
</tr>
<tr>
<td>%Protrusion*%Protrusion</td>
<td>$C_{33}$</td>
<td>0.1454</td>
</tr>
<tr>
<td>%ECB Length*%ECB Width</td>
<td>$C_{12}$</td>
<td>0.0541</td>
</tr>
<tr>
<td>%ECB Length*%Protrusion</td>
<td>$C_{13}$</td>
<td>-0.2521</td>
</tr>
<tr>
<td>%ECB Width*%Protrusion</td>
<td>$C_{23}$</td>
<td>-0.0141</td>
</tr>
</tbody>
</table>

**Summation** 100.0%
CHAPTER 4: DISCUSSION

In order to understand the performance of the diffuser and ECB system, the inlet profile to the system must be well defined. Previous studies have used fully developed inlet conditions (Dunn, Ricklick, & Kapat, 2009) however this study investigates a uniform velocity profile at the diffuser inlet (for the case with no ECB attached). In order to validate our computational model, the ECB was removed from the system, leaving a simple annular diffuser exhausting into the atmosphere. This was done to give access to the exit of the diffuser. During this phase, the inlet and exit velocity profiles were measured with a hotwire anemometer and validated against the computational model with the ECB removed as well. Two traverse locations were chosen at the exit of the diffuser: directly downstream of a strut (7:00), and in-between struts (6:00). These locations were chosen to experimentally define the wake region downstream of the airfoil struts, and compare them to the computational domain. The results for these experiments are shown in Figure 46. The inlet velocity profile was determined to be a flat velocity profile from the experiment and is shown in Figure 44, where the maximum variation in the average velocity through each circumferential location (i.e. 1:00, 2:00, 3:00 etc.) was less than 2 m/s (or approximately 11%). The boundary layer thickness was not measured in the experiment. The single hotwire was able to measure the root mean square of the instantaneous axial velocity fluctuations \( u_{\text{rms}} \), thus the turbulent intensities (TI) were calculated. It should be noted that the turbulence quantities obtained assume isotropic turbulence \( u_{\text{rms}} = v_{\text{rms}} = w_{\text{rms}} \) which is not the case at the...
diffuser exit in this system (El-Behery & Hamed, 2011). Since $v_{rms}$ and $w_{rms}$ are smaller than $u_{rms}$, the actual turbulent kinetic energy is smaller than the isotropically derived quantity. This causes the experimental data to see higher turbulence than the computational results. On top of this effect, the steady RANS model does not model the oscillating eddies which most likely exist in this system (Pope, 2000). These eddies can promote turbulent transport through the diffuser. Therefore, the RANS model incorrectly dissipates the turbulence quantities between the struts, causing low turbulence values in this region. For this reason, the trends of the turbulence (as opposed to the magnitudes) are compared experimentally and computationally by plotting the turbulence intensity (TI) normalized by the local average turbulence intensity ($T_{I_{avg}}$).

Where, $T_{I_{avg}}$ is the average turbulence intensity in the free stream at each measurement location (experimentally and computationally), negating the boundary layer as the experimental data does not extend close to the wall.

The average turbulence intensity at each location is presented in Table 7. As expected, the average turbulence intensities measured by a single hotwire are high (nearly double the computational results), however the trends are more accurate. The diffuser inlet conditions (Figure 46 – bottom) are accurate in both magnitude and trends, as the velocity fluctuations are close to isotropic. The computation correctly captured the trend in turbulence directly downstream of a strut (Figure 46 – mid). An under prediction the turbulence in the high velocity region, and an over prediction of turbulence in the low velocity region is seen.

Between the struts (Figure 46 – top), however, is a poor agreement between experimental and computational results. The computational velocity profile is shifted
towards the center of the annulus, while the magnitudes are accurate within 3%. The minimum turbulence intensity occurs at the location of maximum velocity for both the experiment and computation, however the under prediction in high velocity regions is exaggerated between the struts. Again, this leads to the conclusion that the $k-\varepsilon$ model is incorrectly dissipating the strut wakes, therefore dissipating turbulence between the struts. These results also reiterate the fact that the axial velocity fluctuations ($u_{rms}$) are the dominating factor in the shape of the turbulence intensity curve, as the trends of an isotropically derived quantity and the anisotropic $k-\varepsilon$ turbulence are similar.

It was observed that the flow between the struts creates a velocity profile along the inner annulus that is analogous to the outer annulus. This trend is not seen directly downstream of the struts, as the flow tends to prefer the inner annulus. This is due to a small separation zone on the outer annulus of the diffuser just behind the struts. The geometric diffusion on the outer annulus causes this small separation, while the inner annulus which remains straight has a less significant separation zone. This outer annulus separation zone causes the bulk flow to shift closer to the inner annulus, resulting in a higher velocity. This effect proved to be true by analyzing the computational model.

The performance of the diffuser with no ECB (free discharge condition) was also calculated both experimentally and computationally and shown in Figure 47. It is observed that the computation slightly over predicts the pressure recovery through the diffuser in the low X/L range (maximum difference in Cp of 0.09); however the total amount pressure recovered by the diffuser was within 2% between the computation and the experimental results.
This is caused by an under prediction of the separation on the outer annulus of the diffuser. The reason why the Cp value is not symmetric is due to the fact that the inherent behavior of the diffuser by itself is naturally unsteady. The steady RANS model will not model the unsteadiness of the system, therefore the model will not fully converge to a single solution. The “converged” model has oscillatory residuals which validates the assumption of unsteadiness in the model. The separation within the diffuser moves around slightly at each iteration, which causes a difference in Cp values at certain locations. If an unsteady RANS were to be time averaged over a long period of time, these Cp curves would fall on top of each other due to the symmetry. Figure 47 shows an example of this asymmetric separation bubble within the diffuser. The white zones are reversed flow areas within the diffuser. While this solution may not be physically accurate (due to the reasons stated above) it is an example of a situation in which a symmetric model with symmetric boundary conditions can cause variations in static pressure at the inlet of the diffuser. It should be noted that while the diffuser alone has a moving separation zone, the full computational domain (including the ECB) did not have an unsteady nature, and converged to a single solution. The backpressure effects of the ECB forced separation in specific locations, killing the unsteady oscillations of the separation zone.

By comparing these experimental results with the computational results, it is apparent that the trends were captured correctly. The static pressure is over predicted along the diffuser due to the under prediction of separation, however this effect will be addressed further later. The experimental results describing the inlet velocity profile to the diffuser gives confidence to the inlet boundary condition used in the computational
model as a constant mass flow rate, thus a flat velocity profile. These results give enough confidence in the understanding of the computational model to continue with the main study of this thesis, which is the performance of the entire exhaust system.

Experimental: Inlet Pressure Variation

To build confidence in the inlet condition applied to the computational model, the circumferential static pressure at the OD of the diffuser inlet is shown for each case. Figure 48 shows the static pressure variation from the experiment for Case 1, which is the same ECB geometry used in the inlet conditioning CFD model. Recall that the inlet condition model (described in Chapter 4) calculated a circumferentially asymmetric pressure and velocity field at the inlet of the diffuser. Due to the high pressure zone at the 6:00 region at the diffuser exit, more flow was routed towards the 12:00 region thus lowering the pressure locally. This effect propagates back to the diffuser inlet creating an asymmetric pressure field inlet condition. The experimental results capture this effect perfectly. The variation in the static pressure from 12:00 to 6:00 is over 25% of the inlet dynamic head. In this small scale experiment, this variation is miniscule compared to an industrial gas turbine. The largest gas turbines have a mass flow through the engine of over 500 kg/s, with a turbine exit (diffuser inlet) Mach number of around 0.6. Assuming the turbine exit temperature is at 600 °C, the dynamic head of the hot gas path is around 25 kPa (3.6 psi), thus a 25% variation circumferentially accounts for a pressure difference of about 6 kPa (~1 psi). Compared to the large magnitudes of pressure within an industrial gas turbine this may seem negligible however, if you consider the fact that the last turbine blade is now exposed to an oscillating pressure field as it rotates, high cycle fatigue can come into the picture. The last turbine blade would see a 6 kPa (1 psi)
oscillating pressure with a frequency of 60 Hz (as the engine rotates at 3600 RPM and the period of oscillation is 1 full rotation). While vibratory modes are most likely not an issue at this frequency, as all parts are designed to have natural frequencies that do not correspond to the driving frequency of 60 Hz, high cycle fatigue can still be influenced. At 60 Hz, a turbine blade will go through $1 \times 10^9$ cycles within 1 year (assuming 24/7 operation). Much like repeatedly bending a paper clip until it breaks, even a small magnitude over a large number of cycles can cause damage to a turbine part. This pressure field should be considered when analyzing the vibratory responses and high cycle fatigue analysis of the last stage turbine blade. This pressure variation changes dramatically with the ECB geometry, as the difference between the best and worst cases is over a factor of 3. The maximum inlet Cp variation is tabulated for each experimental case in Table 13. A key observation is that Case 1 provided the highest inlet pressure variation out of all of the experimental cases. This fact sheds new light on the computational boundary condition applied at the inlet. The assumption made at the beginning of this study was that the inlet condition did not vary from case to case. Unfortunately, the experimental data does not back up this assumption. It should be noted that since the inlet pressure was fixed by a boundary condition for all cases, the CFD model does not provide any results for the prediction of this inlet variation. In order to predict this variation with the CFD model, a larger volume similar to the inlet conditioning model would need to be run for each case. It should be noted that this data proves that the inlet condition for all cases other than case 1 are technically invalid, however the remaining results from the CFD model can still be used for comparison purposes.
### Table 13 – Experimental Inlet Cp Variation

<table>
<thead>
<tr>
<th>Case</th>
<th>Maximum Inlet Cp Variation</th>
</tr>
</thead>
<tbody>
<tr>
<td>Case 1</td>
<td>0.270</td>
</tr>
<tr>
<td>Case 8</td>
<td>0.211</td>
</tr>
<tr>
<td>Case 9</td>
<td>0.198</td>
</tr>
<tr>
<td>Case 10</td>
<td>0.174</td>
</tr>
<tr>
<td>Case 13</td>
<td>0.170</td>
</tr>
<tr>
<td>Case 3</td>
<td>0.158</td>
</tr>
<tr>
<td>Case 5</td>
<td>0.146</td>
</tr>
<tr>
<td>Case 11</td>
<td>0.138</td>
</tr>
<tr>
<td>Case 6</td>
<td>0.120</td>
</tr>
<tr>
<td>Case 2</td>
<td>0.113</td>
</tr>
<tr>
<td>Case 7</td>
<td>0.112</td>
</tr>
<tr>
<td>Case 4</td>
<td>0.094</td>
</tr>
<tr>
<td>Case 12</td>
<td>0.089</td>
</tr>
</tbody>
</table>

Looking deeper into how the ECB geometry effects the inlet pressure variation, it is noticed that Case 1 and Case 4 have the same ECB length and width, yet the inlet pressure variation varies by almost 300%. The only geometric change between Case 1 and Case 4 is the diffuser protrusion (0% for Case 1 and 66% for Case 4). Case 2 and Case 3 have the equal ECB length and width as well, with a diffuser protrusion of 0% and 66% for Case 2 and 3 respectively. Interestingly, the inlet pressure variation for Case 3 (66% protrusion) is 38% higher than for Case 2 (0% protrusion), where the inlet
pressure variation for Case 1 (0% protrusion) is 288% higher than for Case 4 (66% protrusion). This leads to the conclusion that the inlet pressure variation is a combination of two (or more) factors, rather than a single variable. At a small ECB width (-50%), increasing the diffuser protrusion will reduce the inlet pressure variation while at a large ECB width (+50%) it will increase the inlet pressure variation. The ECB Length does not seem to have a significant effect on the inlet pressure variation. Interestingly, not only does the magnitude of the circumferential pressure variation change with the ECB geometry, the location of the high pressure zone also changes. The results for Case 3, Case 7 and Case 8 show that the 6:00 location actually has a lower pressure than the 12:00 location (see Figure 56, Figure 72, Figure 76). Each of these cases has a diffuser protrusion of 66%, however Case 4 also has a protrusion of 66% yet does not show this trait. It is observed that all cases that have a reduced ECB width (-50%) show a high pressure zone at the 6:00 region, and all cases that have a 66% protrusion have a high pressure zone at the 12:00 region with exception to Case 4 which has both a reduced ECB width and a 66% protrusion. This leads to the conclusion that a reduced ECB width promotes a high pressure zone at the 6:00 region, and a 66% protrusion promotes a high pressure zone at the 12:00 region. When both geometric traits are included, they combine to form a high pressure zone at the 6:00 and 12:00 regions, with low pressure zones in the 3:00 and 9:00 regions (Figure 60). It should be noted that Case 4 has the second least variation out of all cases tested, and this can be contributed to the combination of the 66% protrusion promoting high pressure at 12:00, and a -50% ECB width promoting a high pressure at 6:00. These two factors tend to even out the inlet pressure profile.
Using the Box Behnken design methodology, a regression equation and response surface could be created using the experimental results for the inlet pressure variation. The commercial software (MINITAB) was employed for the sensitivity analysis, and Table 8 shows the coefficients of each factor in the regression equation using a second order polynomial model shown below. Note that X can be either Inlet Pressure Variation, System Cp, or System Total Pressure Loss, and L, W, and P are the ECB Length, Width, and Protrusion respectively.

\[
X = C_0 + [C_1(L) + C_2(W) + C_3(P)] + [C_{11}(L \cdot L) + C_{22}(W \cdot W) + C_{33}(P \cdot P)]
\]

\[
+ [C_{12}(L \cdot W) + C_{13}(L \cdot P) + C_{23}(W \cdot P)]
\] (4)

The coefficients agree with the physical explanation of the inlet pressure variations, concluding that 33.6% of the contribution comes from the second order effect of the ECB Width and Protrusion factors. Also, the primary factor of the ECB width is the second strongest factor, weighing in at 21%. The Protrusion primary factor contributes less than half of the Width, and the Length contributes over an order of magnitude less than the Width. This regression equation can be used to interpolate results within the design matrix in order to plot a response surface of all three variables. For these plots, two variables are plotted while the third remains constant at the default (0) level. Figure 286, Figure 287 and Figure 288 show the three possible center planes within the 3 variable cube. Imagine a cube from -1 to +1 in the x, y, and z axis, these planes plotted would be the XY plane at Z=0, XZ plane at Y=0, and the YZ plane at X=0. It can be seen that the ECB Width and Protrusion factors provide the largest impact on the inlet pressure variation (concurrent with the large coefficient in the regression equation). The effect where at a small ECB width (-50%), increasing the diffuser protrusion will reduce
the inlet pressure variation while at a large ECB width (+50%) it will increase the inlet pressure variation can easily be seen in Figure 288. To a lesser extent, this phenomenon is also observed in Figure 287 where at small ECB lengths, increasing the protrusion will increase the inlet pressure variation while at larger ECB lengths this effect reverses.

**Experimental System Performance**

![Experimental System Performance Graph]

*Figure 301 - Experimental System Performance Comparison*

The system performance values for each experimental case are shown in Figure 301 in ascending order of system pressure recovery. It is evident that Case 8 and Case 9 are
the worst performing geometries by a large margin. The problem with Case 8 is evident by inspecting the geometry, as the combination of the reduced ECB Length (-50%) and the increased protrusion (66%) leaves only a small gap for the air to pass through when exiting the diffuser. In fact, there is only 65% of the diffuser inlet flow area between the diffuser protrusion and the impingement wall of the ECB. This effectively turns the diffusing system into an orifice-like system where the diffuser exit flow experiences a 4.3:1 contraction ratio immediately after exiting the diffuser, which then dumps into the ECB (see Figure 202 and Figure 204). It is apparent that this is not an ideal system when the goal is to recover pressure. Case 9’s performance can also be contributed to the geometry of the ECB, as the area ratio between the diffuser inlet and the ECB exit is the smallest out of all of the cases studied (due to the -50% ECB Length and Width). The area ratio between the diffuser exit and the ECB exit for this case is nearly unity, giving the flow no extra area to expand after dumping out of the diffuser exit. Additionally, due to the -50% ECB Length alone (even without a diffuser protrusion) the flow exiting the diffuser still has to contract to turn into the ECB by through a contraction ratio of 1.5:1. These two effects combine to create a large pressure loss through the ECB, and stunt the pressure recovery possibility due to the decreased area ratio.

It is observed that four out of the bottom 5 performing cases have a reduced ECB Length, leading to the conclusion that this should be a large contributor to the regression equation for the system Cp. This fact is confirmed in Table 9, as the ECB Length primary factor is the largest weighted contributor at 24%. The third largest factor is the second order term of the ECB Length and Protrusion factors, which can be explained by the extremely poor performance of Case 8. Interestingly, the area of the
ECB exit (ECB Length * ECB Width) is not a strong contributor to the system pressure recovery. This is a strong conclusion, as most designers will take the assumption that a larger area means a higher potential pressure recovery and will design a part with a large area. It should be noted that the best performing case (Case 11) does indeed have the largest ECB cross sectional area, however attention should be drawn to Case 1 and Case 7. Case 7 has double the cross sectional area at the ECB exit as Case 1, however the static pressure recovery of each case is nearly identical. The major difference between these two cases is the Protrusion percentage, as Case 1 has a 0% protrusion and Case 7 has a 66% protrusion. This, combined with the fact that all four cases with a 66% protrusion were in the bottom 50% of the performance rankings, leads to the conclusion that the diffuser protrusion of 66% is unfavorable. From a physical perspective, there may not be a reason why the system Cp reduces when 66% of the ECB Length is taken up by the diffuser protrusion. What is actually happening, is that the diffusing flow is being pinched by the reduced gap between the diffuser exit and the ECB caused by this protrusion. It is theorized that as long as the area ratio between the diffuser exit and the cross sectional area between the protrusion and the ECB impingement wall is greater than 1, the protrusion percentage would not be a large factor. It is just coincidence that in this study the geometric parameters chosen happen to cause an area reduction in this location. It is also noticed that the ECB Width is not a driving factor in any of the first or second order coefficients.

In order to investigate the effects of all three geometric variables, response surfaces can be very helpful. The response surfaces for two variables (holding the third at the nominal (0) value) are shown in Figure 289, Figure 290, and Figure 291. The
largest variations in the system CP are seen in Figure 291 as the ECB Length and Protrusion factors are shown. It is clear that a short ECB Length and a large protrusion is devastating to the system pressure recovery, as discussed earlier. On the other hand it is observed that at a large ECB Length, increasing the protrusion percentage can actually increase the pressure recovery of the system. It can also be reiterated here that the ECB Width is a very minor factor, as both response surfaces with this variable show a minor change in the system Cp when the ECB Width changes. Holding the protrusion at a constant 33%, Figure 289 effectively shows the response surface of the ECB exit area (Length * Width). As stated earlier, this second order term was not a large contributor to the regression equation, which can be seen here in this figure. The ECB Length is a dominating factor, leaving the system Cp almost independent of the ECB Width (indicative of the nearly vertical lines of constant Cp).

The total pressure loss coefficient, or K-factor loss, follows a very similar trend to that of the pressure recovery coefficient. In fact, from the experimental results the response surfaces (Figure 292, Figure 293, and Figure 294) are nearly identical to the pressure recovery response surfaces. The difference between the system pressure recovery and the total pressure loss can be attributed in part to the turbulent mixing losses within the system. The coefficients on the regression equation are similar, however there are a few minor differences between the system Cp and total pressure loss equation coefficients. The total pressure loss coefficient for the ECB Length dropped by a half a percentage point, while the Protrusion increased by 0.7 percentage points. Similarly, the second order ECB Length * Protrusion increased by 0.6 percentage points. These variations are small enough to be considered noise, as the
curve fitting regression equation, as well as the experimental uncertainty, is within this order of magnitude.

**CFD System Performance**

The goal of this study from a computational perspective was to determine the accuracy of the industry standard CFD tools from a system performance standpoint. The question that is to be answered in this study is whether or not the CFD study provides the same conclusions as the experimental results. In industry, it may be determined that a design choice will be based only on computational results rather than experimental results for multiple reasons. Generally, CFD can provide results in a matter of weeks where an experimental study could take months. Additionally, for small scale or short term studies, CFD tends to be less expensive as it requires minimal man-hours of setup and no machining costs (for larger scale and/or more complicated studies, experimental tends to be cheaper). The decision to use CFD as a design tool rather than experimental results is heavily dependent on the complexity of the physics of the problem, and whether or not the computational code can accurately represent the flow field. Using CFD to predict 3-dimensional adverse pressure gradient systems has been under scrutiny for many years, as the scientific community has not yet obtained a full and complete understanding of the behavior or these systems. This study has been conducted to determine if the correct design decision can be made using only CFD results.
The system performance values for the CFD cases are shown in Figure 302 in ascending order of system pressure recovery. As with the experimental results, Case 8 and 9 are the worst performing systems. Again, this is due almost entirely by the geometric restrictions on the airflow path at the exit of the diffuser. Moving on to Case 4 and 12, it is noticed that the order of these two cases is switched between the experimental results and the CFD results. While the CFD does agree that both cases are relatively poor in performance, the CFD predicts that Case 12 has higher Cp (+0.15) and lower total pressure loss (-0.16) relative to Case 4. The experimental results show
that Case 12 has lower $C_p$ (-0.08) and a higher total pressure loss (+0.1) relative to Case 4. The differences between the experimental and computational results can be contributed to a number of factors. First, these two cases have a reduced ECB area combined with a non-zero protrusion. These effects lead to high velocity regions within the ECB, including high energy vortices which promote high turbulence. It has been concluded in previous studies (Bernier, Riclick, & Kapat, 2011) that these systems are difficult for the CFD to accurately predict the pressure losses. Second, the inlet condition of the CFD model was not accurate as discussed previously in Chapter 4. The inlet condition applied forces a high pressure at the 6:00 region, which promotes more flow through the upper half (12:00 region) of the diffuser. Looking at the results from the experimental inlet static pressure (Figure 92), it is clear that the inlet pressure is nearly uniform, varying between +/- 0.05 times the dynamic head. Therefore, in the experimental rig, the amount of mass flowing through the 6:00 region is nearly equivalent to the mass flowing through the 12:00 region. Looking at the CFD results of the diffuser exit velocity profile (Figure 263) for Case 12, it is noticed that a large portion near the 6:00 region has separated off of the diffuser OD due to the restriction in area in the ECB at that location (see bottom of Figure 262). This region has a large amount of total pressure loss due to the separation and recirculation which propagates all the way back to the inlet of the diffuser. Since the experimental measured more flow through this region of the diffuser, it is applicable to make the assumption that there would be a higher total pressure loss through the 6:00 area than in the CFD prediction, which has less mass in the 6:00 region. Comparing these effects to Case 4, it is noticed that the experimental inlet pressure variation is much closer to that of the applied inlet condition.
to the CFD model, showing a high pressure region at 6:00. This leads to the conclusion that the CFD and experiment should be experiencing the same physics, as the inlet condition should be very similar between the two. Therefore, the CFD should conclude that Case 12 has a lower total pressure loss than Case 4 (due to the reduced flow in the high-loss 6:00 region), where the experiment should conclude the opposite (due to having more flow in the high-loss 6:00 region).

Continuing the comparison on the system performance values, it is noticed that for the remainder of the cases the CFD agrees on the order of performance for each case. It is noted that the order of Case 6 and 10 is switched, however the difference between both cases was within the experimental and computational uncertainty and can be considered equivalent. In order to fully understand what the CFD results show, it is helpful to look once again at the Box-Behnken design response surfaces. Comparing the system C_p, it is clear that the CFD response surfaces are extremely similar to the experimental results. The CFD agrees that when holding the Protrusion constant, the ECB Length is a dominant factor over the ECB Width as the slope on the constant C_p curves in Figure 295 are nearly vertical. Similarly, at a constant ECB Length shown in Figure 296 it is noticed that the ECB Width is once again a minor variable compared to the Protrusion, however it is less of a difference compared to the ECB Length. From those two observations, one can conclude that of the primary variables in the regression equation the ECB Length should be the heaviest weighted, followed by the Protrusion then the ECB Width as the least weighted variable. These observations are confirmed by investigating the regression coefficients shown in Table 11. Looking at Figure 297 and Figure 291, it is noticed that the slope of a constant C_p line is nearly identical in
both cases. This lead to the investigation on the reasoning behind this slope, as it was stated earlier in Chapter 4 that the area ratio between the diffuser exit area and the flow are between the Protrusion and the ECB is an important factor. Shown in Figure 303 is the areas in question (on the Right) along with a line of constant area ratio equal to 1 plotted on the response surface of the ECB Length and Protrusion factors. It is noticed that at high Protrusion percentages, this area ratio line is parallel with the lines of constant Cp and nearly located at the maximum value of Cp. It can easily be concluded that if the diffuser protrudes into the ECB by more than 30%, a maximum Cp value of the system can be obtained by setting the area ratio (A1/A2) equal to 1, with little benefit from increasing this area ratio. Interestingly, for a system with no diffuser protrusion this conclusion is not valid. It is observed that A2 needs to be 50% larger than A1 before reaching the optimum system Cp. This leads to the conclusion that there are other effects within the ECB that are reducing the pressure recovery of the system. These effects could be the formation of the vortex at the 6:00 location in the ECB, and how it forces separation off the diffuser OD (see Figure 104). Increasing the protrusion percentage helps to block this effect from happening by separating the ECB 6:00 region with the diffuser 6:00 region.
Figure 303 - Area ratio definition between the Diffuser exit (A1 revolved around the Centerline) and the area between the Protrusion and the ECB wall (A2 revolved around the Centerline)

Similarly, the total pressure loss response surface also follows the same behavior with respect to this area ratio line both experimentally (Figure 294) and computationally (Figure 300). As with the experimental total pressure loss response surfaces, the computational results are nearly equivalent in trends to the system pressure recovery. The only minor difference coming between the ECB Width and Protrusion response surface (Figure 290 experimentally, Figure 296 computationally) at low ECB Width values. This difference is explained by the difference in Cases 4 and 12 explained previously in this section. Case 4 from the computational results shows a higher pressure loss than Case 12, due to the incorrect inlet boundary conditions, which drives the bottom left corner of this response surface towards the higher end of the pressure loss.
CHAPTER 5: GENERAL DISCUSSION/CONCLUSIONS

The technology and tools used in the gas turbine industry have been increasing in complexity and accuracy over the last few decades. Engineers are relying on computer models and correlations more and more in order to predict the performance of a typical engine part. To this end, academic studies have flourished with a main focus on investigating the accuracy of these models, where they work, and most importantly where they do not. Human beings have been using tools for thousands of years, all with a common purpose: to make our jobs easier. Many years ago, these tools could have been hard rocks which were used to smash into other rocks in such a way as to make sharp points. These points would then be tied to sticks, and could be used as weapons allowing us to hunt larger animals and provide enough food to live comfortably. More recently, these tools could have been an ox-driven plow allowing us to cultivate more land, which would allow for more crops and eventually lead to a more comfortable life during harvest season. Today, these tools still exist although in some cases they are no longer tangible. CFD is a tool used in industry, as a way to predict how well (or poor) a certain design will perform. This tool allows us to design better engines that produce more power while using less fuel, which in turn allows us to harness this power in our daily lives (making our average day easier) while still keeping a close eye on our environmental health. While this last example is somewhat of a stretch, the idea is still sound. The only difference between the tools that our ancestors used and the tools that
we use today are the complexity of the tool itself, not necessarily what the tool is intended to do. Using a rock to make an arrow head is simple, and there are not very many ways that it could go wrong. An ox-driven plow is slightly more complex, however the tool can still only do what it was built to do. Today, however, tools such as CFD are so complex that many of the people who are using it don’t even know what it does. The complexity of the tool has made it such that the inaccuracies of the tool in certain scenarios may outweigh the benefit that the tool gives us as engineers. Unfortunately, it is not a simple task to answer the questions such as “Under what conditions does the tool work?” or “When won’t the tool work?” Often, expensive and long term studies must be conducted (such as this one) in order to determine how the tool works, where it works well, and where it works poorly. The answers these tools give us are not worth the paper they are printed on (or more accurately: the hard drive space in which they are stored) unless the tool can be trusted. The only undeniable way to validate a tool is to test it, and that is exactly what this study has done.

In this study, a very common industry tool (CFD) is tested under a notoriously harsh environment: three-dimensional adverse pressure gradient separated systems with recirculation and high vorticity. Many years of work have gone into these CFD tools trying to design them to predict these separated flows accurately. Only recently have these tools been accurate enough to be used to predict such complicated flows, although they are never perfect. The goal of this study was to use this tool (steady RANS with the $k-\varepsilon$ turbulence model) to predict the performance of an exhaust diffuser system under a wide variety of geometric variations. The predictions were evaluated on two different levels, locally and globally. Locally, the computational code was validated
by comparing pressures at specific locations within the diffuser and ECB. Globally, the
code was validated using system performance numbers such as system Cp and total
pressure loss. These two different levels of validation are used to determine not only if
the CFD can predict how well a design is performing, but whether or not it predicts ‘why’
it is performing. More often than not, the answer to ‘why’ something works is more
important (and much more interesting) than ‘does’ something work. Learning whether or
not a specific design works can’t help you change the outcome, it can only tell you
which one is better. Knowing how and why each design is different allows you to
manipulate the physics to get the answer that you want (or at least closer to it than you
were before).

In order to determine whether or not a steady RANS with a k-ε turbulence CFD
model can accurately predict the flow field and performance of a gas turbine exhaust
diffuser, two identical studies were run. A total of 13 geometric variations were set up in
such a way that a Box Behnken design could be used to create a response surface of
three geometric variables (ECB Length, Width, and Protrusion). These 13 geometries
were built and tested in the Center for Advanced Turbines and Energy Research, a
Laboratory for Turbine Aerodynamics and Heat Transfer at the University of Central
Florida. A small scale wind tunnel was designed and built specifically for this experiment
with a controllable inlet condition, and an exhaust into the atmosphere. The
performance of each case was measured using static pressure ports and aerodynamic
probes such as a Kiel head probe and a hotwire anemometer. In parallel with this study,
a purely computational study was conducted using the exact same 13 geometric
variations, with boundary conditions mapped on from inlet condition models. These two
experiments were run and processed separately, meaning that the computational model uses no data from the experiment, and the experiment uses no data from the computations. This was done to evaluate each experiment as a stand-alone test in an attempt to mimic what a typical industrial application would be, as experimental and computational data on a single component is often rare, generally it is one or the other. The comparison of each experiment type is shown in Figure 304 and Figure 305.

Figure 304 - Experimental and Computational comparison of the system Cp
It was noted previously in Chapter 4 that the computational model accurately predicts the order (best to worst performing with respect to Cp) of the cases very accurately (see Figure 304). The only discrepancy comes between Case 4 and Case 12 which was determined to be a combination of an unrealistic inlet boundary condition and increased energy within the system, both of which contribute to poor computational performance. It has been concluded previously in literature (Bernier, Riclick, & Kapat, 2011) that high energy vortices tend to stunt the accuracy of computational models, while low energy/slow moving flows are much easier to predict. Combine this with the knowledge that due to the experimental Cp normalization procedure producing higher than real Cp values, it can be concluded that for the majority of the cases (all other than
Case 8, 9, 12) are predicted very accurately. If one were to use the experimental results alone, the conclusions on system performance would be identical to those made by using the CFD results alone. As for the total pressure loss coefficient (Figure 305), similar conclusions can be made. The worst performing cases (Case 8, 9, and 12) had some of the highest velocities within the ECB due to a reduced length (-50%) and a non-zero protrusion percentage which acted as a pinch point in the flow. These high velocity regions consist of strong vortex structures which contribute a significant amount of pressure loss to the system. The pressure loss that comes from these vortices is not easy to predict using a steady RANS solution, as these vortices are not resolved. The “average” predictions of the RANS solutions do not accurately predict the flow behavior within a vortex structure. This is one of the two main reasons why Case 8, 9, and 12 are both the worst performing cases, and why they have the worst agreement between experimental and computational results. The second reason is due to inaccuracies in the experimental measurements. The exit total pressure profile of the ECB was measured a 0.375” diameter Kiel probe which was traversed to between 60 and 100 locations at the ECB exit (dependent on ECB exit size). It is noticed from the CFD results, that the ECB exit total pressure profile is far from uniform. In fact, there are small regions in which a large portion of the exit total pressure exists (see Figure 212 for Case 8). It was concluded that the spatial resolution of the exit profile mapping from the experimental rig was insufficient to capture the small regions of high total pressure. Therefore, the total pressure at the exit of the ECB which was measured by the experiment is noticeably lower than the computational prediction, granting a larger total pressure loss coefficient. This effect is strongest when the ECB exit total pressure is
high, which occurs in the cases with the smallest ECB exit area (Case 5, 8, 9, 12). It is not a coincidence that the largest variation between experimental and computational results occurs on these 4 cases. Once the ECB exit area increases, the average total pressure at the ECB exit drops (by nearly a 1:1 ratio), therefore the errors in the experimental measurement are not nearly as significant as before. The cases with the largest ECB exit area (Case 2, 6, and 11) show the best agreement between the computational and experimental total pressure loss coefficients, showing an error of only 7% for Case 2 and 6, and less than 5% for Case 11. Another interesting note, is the effect of the inlet boundary condition applied to the CFD model. The inlet conditioning model that was created to estimate the inlet velocity profile used Case 1 as the ECB geometry. The assumption was made prior to conducting this study that this inlet boundary condition would not change with geometric variations of the ECB. It was concluded after the study, that this assumption was invalid, as the effects of the ECB are strong enough to change the pressure field at the diffuser inlet. It just so happens that Case 1 had a perfect agreement between the experimental and computational results on the total pressure loss of the system. Unfortunately, this does not mean that the computational model is perfect. While it is true that Case 1 has the most accurate inlet boundary condition out of any of the cases, it does not mean that the errors in the computational code and the experiment go away. The experiment still may not be capturing the ECB exit total pressure profile exactly, and the computational code may still not be accurately predicting the pressure losses through the vortex structures within the ECB. It is only coincidence that the numbers ended up canceling out, granting a near perfect agreement between the computational and experimental results. To
understand this further, one can look at the local pressure data on the diffuser OD and the ECB walls. It is noticed that the location of the vortex core is predicted very well with the computational code, as the minimum pressures along the ECB wall occur at the same location (about 23% ECB Width) in the experiment (Figure 50) and in the CFD (Figure 110). Additionally, the interaction between the vortex and the side wall is captured as well, as the inflection point in the static pressure for all 4 rows occurs at 12% ECB Width experimentally and computationally. The differences occur at the prediction of the magnitude of the vortex core pressure. The CFD predicts that the core of the vortex shows a pressure about 0.2 times the dynamic head lower than the average exit pressure, while the experiment shows half of this magnitude. This fact alone enforces the conclusion that the CFD is not perfectly predicting the flow within the ECB. Weight should not be taken away from the conclusion that a proper inlet boundary condition to a diffuser computational model will provide the most accurate results, but bear in mind that it will not provide perfect results.

The conclusions from this study can be summarized as follows:

- Geometric variations in the ECB will change the pressure field at the inlet of the diffuser. This effect will change the inlet velocity profile to the diffuser as well, and should be accounted for during computational studies. It is suggested that caution should be used in industry when applying boundary conditions obtained from upstream components, such as the last turbine blade in this scenario. If the exit profile of the turbine is used as the inlet condition of the exhaust diffuser model, errors could occur when changing the diffuser geometry without changing the inlet condition. In fact, it is highly suggested that the exhaust diffuser should
be an integral part of the last turbine stage model whenever possible to avoid this boundary condition issue.

- The system pressure recovery and total pressure losses are dominated by the ECB Length and Protrusion factors. In general, increasing the ECB Length will increase the pressure recovery and reduce the total pressure loss, while increasing the Protrusion will reduce the pressure recovery and increase the total pressure loss, albeit at a much slower rate than the ECB Length changes. The effect of the ECB Length * Protrusion factor is very strong under the condition that the area ratio between the diffuser exit and the Protrusion to ECB gap is below 1. It is highly suggested that the area ratio should be designed to be at or greater than 1 in all cases, as diffusing systems are not meant to have contracting areas within them. Once the area ratio is greater than 1, the ECB Length * Protrusion factor becomes very weak. The diffuser inlet pressure variation can be minimized by increasing the ECB Width or increasing the Protrusion.

- The realizable k-ε turbulence model is an accurate tool to determine if one design is better than another. The CFD code was able to accurately predict the correct order of performance for nearly all 13 cases, even without using accurate inlet boundary conditions. The agreement between the CFD and experimental system total pressure losses were within 10% for the high volume/low energy cases (Cases 2, 6, 11), and the trends in the system pressure recovery for all cases other than Cases 8, 9, and 12 were accurately described by the CFD model. Locally, however, the computational code was the most accurate for the Case 1
due to the correct inlet boundary condition. It is suggested that the inlet boundary of any exhaust model should be placed far away from the diffuser inlet so that the back pressure effects of the ECB can propagate upstream and create the correct boundary condition for the diffuser. With a general inlet boundary condition, the CFD model was determined to be accurate for case to case comparisons, with a true inlet condition the CFD model was determined to be accurate locally as well.
CHAPTER 6: FURTHER STUDIES AND RECOMMENDATIONS FROM LESSONS LEARNED

Significant effort was taken to create a flat velocity profile at the inlet of the diffuser. The design iterations of the plenum design and flow distribution system are shown here.

Splash plates

Three splash plates were placed at the center of the bottom wall of the plenum. The incoming flow from the pipe in the side wall impinges onto these three plates, which distributes the flow around the plenum. This enabled us to run a very high flow rates, as the pressure drop from these splash plates was very low. However, the flow distribution was far from even at the entrance to the diffuser.

- Low pressure Drop
- Uneven Flow Distribution
These splash plates created a strong fluctuation in the velocity profile at the end.
of the nozzle, which caused a huge variation at the end of the diffuser. It was obvious that there was some complex reaction to the flow impacting these splash plates, and decided to look further into it. The experimental results are shown below.

Quantitatively, the maximum gage total pressure variation (max to min) at the exit of the nozzle was 12%, corresponding to a velocity fluctuation of 5.9%. At the exit of the diffuser, these were drastically exaggerated, causing a pressure variation of 950% and
a velocity fluctuation of 225%. After seeing these results, we felt the need to look deeper into the cause of this issue and look for ways to fix it. The CFD results are quite interesting when compared to the experimental results. The streamlines indicate a significant fluctuation at the exit of the plenum, with the majority of the flow on one side of the exit. Looking at the total pressure profile at the exit of the diffuser, this is also evident. In fact, the same side of the diffuser is experiencing the increase in flow as in the CFD prediction.

**PVC Ducts**

A quick and dirty test to see how the flow would be distributed from discrete holes placed in a series of PVC pipes inside the plenum. The incoming flow from the blower was directed through a series of PVC pipes laying at the bottom of the plenum. The exits of all of these pipes were capped shut, and holes were drilled into the sides and top of each PVC pipe for the flow exit. The flow distribution was excellent, however the pressure drop was incredibly high, lowering our potential flow rates for the experiment.

- High Pressure Drop
- Even Flow Distribution
Figure 310 - Discreet holes placed in PVC pipes at the bottom of the plenum

Figure 311 - Gage total pressure variations circumferentially for the exit of the nozzle
Figure 312- Gage total pressure variations circumferentially for the exit of the diffuser

Quantitatively, the nozzle exit pressure variations were around 10%, resulting in a velocity variation of 5%. For the exit of the diffuser, the circumferential pressure variation was as much as 180%, resulting in a velocity fluctuation of 68%.

Clearly, this design is much more effective in distributing the flow evenly through the nozzle, which results in a diffuser exit profile which is closer to normal than the original design. The drawback of this design is the massive pressure loss through the PVC sections, using the original splash plate design it was possible to reach just over 500 CFM through our experiment, however with this design we lost 20% of that, reaching only 400 CFM. This design was not adequate, as a limit of 400 CFM on the maximum flow rate will cause difficulty in the future experiments. Further design iterations were required in order to design a working prototype to be used in the rig for the remainder of the experiments.

Rapid Prototyped Pressure Taps
The diffuser, created by Mydea Technologies, was designed with 16 pressure taps (4 axial taps at 4 circumferential locations). Due to the size of the model (0.45m long including extended inner annulus) the build was split into multiple sections, which were then sealed together in post processing. To complicate the build further, all internal pressure lines that were designed into the model would need to be cleaned out during post processing as well. The tools necessary to clean out the internal lines are unable to navigate around a 90° corner. In order to clean out the internal lines that have more than one 90° bend, the diffuser would have to be split along the centerline of the pressure tap line, cleaned, then sealed back together. This process increases the complexity of the part, as well as the cost.

Figure 313 - Internal pressure lines at the exit of the diffuser’s outer annulus

Internal Vein Analysis
During the post-processing of the diffuser, errors were made regarding the internal pressure lines. Through extensive testing and experimentation, it was determined that the internal lines were leaking slightly to atmosphere. Further testing concluded that the inner annulus ports were also leaking between one another. During the post processing, when the diffuser (in 15 sections) is pieced back together, the sealant used must not have completely sealed the gaps between the internal pressure lines and the outside edge of the diffuser. These errors caused the pressure readings from these ports to be significantly off of what is expected. It is estimated that nearly 75% of the internal pressure lines had some leakage in them.

This issue was solved in a number of ways. The first being external application of epoxy along each of the pressure lines, as well as each section seal. This solved the issue of the internal ports leaking to atmosphere, but it did not solve the internal leakage between lines. In order to avoid these issues, the inner annulus ports were sealed shut and were not used in this experiment. The outer annulus ports that still had remaining leakage issues were filled with epoxy, and another pressure tap was drilled straight through the wall in a location close to (but not on top of) the previous port. All of the pressure taps were validated by drilling a separate tap close to the questioned port and comparing pressures between those ports. Once validated, the extra port was sealed shut and the original manufactured ports were used for the experiments.

For pieces created using the Objet processes from Mydea Technologies, it should be noted that internal pressure lines that do not go straight through a model are extremely difficult for them to manufacture correctly. Taps that go straight through are very easy and work very well.
Surface Roughness Analysis

- **FDM:**
  - The surface roughness was significant in these items, however the lower quality was the result of a far cheaper model. These pieces were chosen to be made out of FDM as they were not part of the measuring section of the flow, only the conditions section.

- **Objet:**
  - The diffuser was created in Mydea’s Objet machine, the surface roughness was negligible, and was considered to be hydrodynamically smooth.

- **Optical Window:**
  - The optical window that was placed into the model had a small lip on the inside of the diffuser, it was upgraded by using putty and sandpaper to smooth the transition between pieces.

Exhaust Collector Box
Construction of the first geometry was made from foam block. The foam was cut with a hot-wire cutter, configured similarly to a band saw. The method of fabrication was the main appeal for using foam, and the notion that rapid test section production and/or alteration was very attractive. Once implemented, the error inherent to marking and cutting the foam by hand became apparent as the smoothness of the foam wall was deemed unacceptable. The foam was then sanded down in an attempt to knock down the irregularities in the wall, but two issues quickly arose; issue number one was that due to their modular nature, sanding one module of the foam geometries meant sanding
all modules that followed. This predicament led directly to issue two, which was the challenge of having each foam module match very well. It was suspected that any variance in the wall angle from the normal would create an unintended trip in the flow path.

In an attempt to salvage the foam effort, it was decided that with equal lengths of coated paper, and so long as the width dimension (relative to exhaust direction) was the same for each module set, could be attached in a manner such that the paper only loosely depended on the foam for shape. In other words, the paper’s main attachment points would be the known-correct exit walls and the shape would predominately be a function of its length. This technique was able to be used in test performance and yielded consistent data, however, the low durability of the paper resulted in structural deterioration. With this, the test geometries were no longer modular and reusable.

After the lifespan of the paper-coated foam was determined to be short, it was decided to try fabricating the geometries with wood. A jig was constructed from a mold that reflected the voids that made up the different test geometries, with the intention of bending the wood about the jig to create the wall of the test section. The process consisted of applying glue to thin strips of wood, then simultaneously layering and clamping those strips around the jig. While the glue was left to set, a frame was fabricated to hold the dried wooden shapes. The hope of a durable, modular test section died with this attempt; the slats were unpredictably challenging to bend with much precision, and the curved sections experienced positional variances as large as one-quarter of an inch.
The final and successful attempt at a uniform and smooth wall came when a thin layer of lexan (similar to flexible plexiglass) was laid over the wooden geometry. The material was extremely smooth, and the materials semi-rigid characteristics eliminated interaction between the flow underlying wooden defects.

Possible Improvements

Improvements to geometry fabrication are abundant. For the foam method, a higher level of precision in the cutting process is a must. Therefore, a method such as hot-knife CNC would be appropriate. Additionally, a heavy coating on the wall surface that would form a smooth surface would solve any surface roughness issues inherent to the foam. Note that a non-caustic coating would be necessary to prevent damage to the foam.

As for the wooden method, the precision of the jig was adequate. The complications arose when the wooden slats were glued and formed around the jig. The process needed to be completely quickly and precisely, and also required a familiarity with the technique. Given enough time and practice, the wood forms could be viable. Therefore, improvements would come about via plenty of practice or by hiring a carpenter to perform the work. The final products would be very durable, long lasting test modules. Additionally, a wooden test section would allow for minor modification, such as the addition of pressure taps, to be performed by relatively unskilled personnel.

When all three techniques are compared, the lexan offers the best time to result ratio- by a wide margin. The lexan is as durable as wood, if only a little less forgiving when modifications are performed. In addition, the surface of the lexan is smoother than
the foam, paper, and wood. Therefore, the lexan / wood frame combination is an excellent choice for initial phase testing.
APPENDIX A: UNCERTAINTY ANALYSIS
Instrument Uncertainty

Micro-manometer:
$u_{\text{mano}} := 0.05 \text{ Pa}$

Transducer

$FSR_{t2} := 1.4 \text{ in}_\text{water} \quad Res_t := 0.0025$

$u_{t2} := FSR_{t2} \cdot Res_t$

$u_{t2} = 3.5 \times 10^{-3}$

$u_{\Delta P} := u_{t2}$

Venturi

$FSR_v := 500 \quad Res_v := 0.01$

$u_v := FSR_v \cdot Res_v \quad u_v = 5$

Thermocouples

$u_{\text{th}} := 1 \text{ K}$

Calipers

$u_c := 0.00005 \text{ (m)}$

$u_{Do} := u_c$

$u_{Di} := u_c$
$T_{in} := 305 \ \text{K}$

$P_{in} := 101325 \ \text{Pa}$

Density measurement:

$R = 287 \ \text{J/Kg*K}$

$\rho(R, T_{in}, P_{in}) := \frac{P_{in}}{R \cdot T_{in}} \quad \rho_{in} := \rho(R, T_{in}, P_{in})$

$u_p := \sqrt{\left( \frac{d}{dT_{in}} \rho(R, T_{in}, P_{in}) \cdot u_{th} \right)^2 + \left( \frac{d}{dP_{in}} \rho(R, T_{in}, P_{in}) \cdot u_{ mano} \right)^2}$

$u_p = 3.795203 \times 10^{-3}$

$\frac{u_p}{\rho(R, T_{in}, P_{in})} \cdot 100 = 0.327869$

Venturi Uncertainty

$DP := 47$

$T := 80$

$P := 0.1$

$\text{Flow}_{\text{GPM}}(DP) = 10 \left( \frac{(\log(DP) + 2.57622)}{2} \right)^2$

$\text{Flow}_{\text{GPM}}(DP) = 133.094138$

$Tc(T) := \sqrt{\frac{T + 460}{330}} \quad T \text{ must be in deg F}$

$Pc(P) := \sqrt{\frac{14.73}{P + 14.73}} \quad P \text{ must be in psi}$
\[
\text{Flow}_\text{SCFM}(\text{Flow}_{\text{GPM}}, \text{T}_e, \text{P}_e) = \frac{\text{Flow}_{\text{GPM}}}{\text{T}_c - \text{P}_c}
\]

\[
\begin{align*}
\text{u}_{\text{Flow}_{\text{GPM}}} & = \left( \left( \frac{d}{d\text{P}} \text{Flow}_{\text{GPM}}(\text{DP}) \right) u_{12} \right)^2 \\
\text{u}_{\text{T}_e} & = \left( \frac{d}{d\text{T}} \text{T}_e(\text{Te}) \cdot u_{12} \right)^2 \\
\text{u}_{\text{P}_e} & = \left( \frac{d}{d\text{P}} \text{P}_e(\text{P}) \cdot u_{12} \right)^2
\end{align*}
\]

\[
\text{u}_{\text{Flow}_{\text{GPM}}} = 4.955533 \times 10^{-3}
\]

\[
\text{u}_{\text{T}_e} = 9.346203 \times 10^{-4}
\]

\[
\text{u}_{\text{P}_e} = 1.176055 \times 10^{-4}
\]

\[
\text{Q}_\text{gpm} = \text{Flow}_{\text{GPM}}(47)
\]

\[
\text{T}_a = \text{T}_c(80)
\]

\[
\text{P} := \text{P}_c(0.1)
\]

\[
\text{u}_{\text{Flow}_{\text{SCFM}}} = \sqrt{\left( \frac{d}{d\text{Q}_\text{gpm}} \text{Flow}_{\text{SCFM}}(\text{Q}_\text{gpm}, \text{T}, \text{P}) \cdot \text{u}_{\text{Flow}_{\text{GPM}}} \right)^2 + \left( \frac{d}{d\text{T}} \text{Flow}_{\text{SCFM}}(\text{Q}_\text{gpm}, \text{T}, \text{P}) \cdot \text{u}_{\text{T}_e} \right)^2 + \left( \frac{d}{d\text{P}} \text{Flow}_{\text{SCFM}}(\text{Q}_\text{gpm}, \text{T}, \text{P}) \cdot \text{u}_{\text{P}_e} \right)^2}
\]

\[
\text{u}_{\text{Flow}_{\text{SCFM}}} = 0.469648
\]

\[
\frac{\text{u}_{\text{Flow}_{\text{SCFM}}}}{\text{Flow}_{\text{SCFM}}(\text{Q}_\text{gpm}, \text{T}, \text{P})} = 100 \times 0.091416
\]

\[
\frac{11}{100} \text{Flow}_{\text{SCFM}}(\text{Q}_\text{gpm}, \text{T}, \text{P}) = 5.530219
\]

Velocity at throat uncertainty
\[ V(Q, A, \rho) := \frac{Q}{\rho \cdot A} \]

\[ Q := 0.269 \text{ kg/s} \]
\[ u_Q := \frac{1.1}{100} \cdot \text{Flow\_SCFM}(Q_{gpm}, T, P) \cdot 0.0005378 \]

\[ D_{\text{out}} := 0.08558212 \]
\[ D_{\text{inside}} := 0.05781382 \]

\[ A(D_{\text{out}}, D_{\text{inside}}) := \frac{\pi}{4} \left(D_{\text{out}}^2 - D_{\text{inside}}^2\right) \]

\[ A(D_{\text{out}}, D_{\text{inside}}) = 0.012509 \]

\[ u_A := \sqrt{\frac{d}{dD_{\text{out}}} A(D_{\text{out}}, D_{\text{inside}}) \cdot u_c}^2 + \left(\frac{d}{dD_{\text{inside}}} A(D_{\text{out}}, D_{\text{inside}}) \cdot u_c\right)^2 \]

\[ u_A = 1.622316 \times 10^{-5} \]

\[ \alpha := A(D_{\text{out}}, D_{\text{inside}}) \]
\[ A = 0.012509 \]
\[ \rho := \frac{101325}{287.305} \]

\[ \rho = 1.157537 \]

\[ u_{\text{velocity}} := \sqrt{\left(\frac{d}{dQ} V(Q, A, \rho) \cdot u_Q\right)^2 + \left(\frac{d}{dA} V(Q, A, \rho) \cdot u_A\right)^2 + \left(\frac{d}{d\rho} V(Q, A, \rho) \cdot u_\rho\right)^2} \]

\[ u_{\text{velocity}} = 0.215589 \]

\[ \frac{u_{\text{velocity}}}{V(Q, A, \rho) \cdot 100} = 1.160499 \]
uncertainty in reynolds number at throat

\[ D = D_{\text{out}} - D_{\text{inside}} = 0.055537 \]

\[ \mu = 1.79 \cdot 10^{-5} \]

\[ \lambda = \frac{0.235}{\rho \cdot A} \]

\[ \text{Re}(\rho, V, D, \mu) = \frac{\rho \cdot V \cdot D}{\mu} \]

\[ u_{\text{Re}} = \sqrt{\left( \frac{d}{d\rho} \text{Re}(\rho, V, D, \mu) \cdot u_{p} \right)^2 + \left( \frac{d}{dV} \text{Re}(\rho, V, D, \mu) \cdot u_{\text{velocity}} \right)^2 + \left( \frac{d}{dD} \text{Re}(\rho, V, D, \mu) \cdot u_{c} \right)^2} \]

\[ u_{\text{Re}} = 799.22002^2 \cdot \frac{u_{\text{Re}}}{\text{Re}(\rho, V, D, \mu)} \cdot 100 = 1.371223 \]

Pressure Recovery Coefficient

\[ p_1 = 145.6 \]

\[ p_2 = 18.3 \]

\[ \rho = 1.157537 \]

\[ V = 16.229199 \]

\[ C_p(p_1, p_2, \rho, V) = \frac{p_1 - p_2}{\left( \frac{1}{2} \right) \cdot \rho \cdot V^2} \]

\[ C_p(p_1, p_2, \rho, V) = 0.835082 \]

\[ u_{c_p} = \sqrt{\left( \frac{d}{dV} C_p(p_1, p_2, \rho, V) \cdot u_{\text{velocity}} \right)^2 + \left( \frac{d}{d\rho} C_p(p_1, p_2, \rho, V) \cdot u_{p} \right)^2 + \left( \frac{d}{d\rho} C_p(p_1, p_2, \rho, V) \cdot u_{c} \right)^2 + \left( \frac{d}{d\phi_1} C_p(p_1, p_2, \rho, V) \cdot u_{c_2} \right)^2} \]

\[ u_{c_p} = 0.022355 \]

\[ \frac{u_{c_p}}{C_p(p_1, p_2, \rho, V)} \cdot 100 = 2.6768959 \]
APPENDIX B: EXCEL SPREADSHEETS
# Ambient Ports

<table>
<thead>
<tr>
<th></th>
<th>Bottom Wall Set</th>
<th>Top Wall Set</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>2</td>
<td>3</td>
</tr>
<tr>
<td>4</td>
<td>5</td>
<td>6</td>
</tr>
<tr>
<td>7</td>
<td>8</td>
<td>9</td>
</tr>
<tr>
<td>10</td>
<td>11</td>
<td>12</td>
</tr>
</tbody>
</table>

-0.00176 -0.0457 -0.16288 0.01156 -0.05229 -0.047953 0.029795 0.038634 0.138653 0.041032 -0.08682 0.079422 -0.06626 0.074138 0.080422 -0.05648

# Inlet Extension Static Pressure

<table>
<thead>
<tr>
<th></th>
<th>1</th>
<th>2</th>
<th>3</th>
<th>4</th>
<th>5</th>
<th>6</th>
<th>7</th>
<th>8</th>
<th>9</th>
<th>10</th>
<th>11</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>2</td>
<td>3</td>
<td>4</td>
<td>5</td>
<td>6</td>
<td>7</td>
<td>8</td>
<td>9</td>
<td>10</td>
<td>11</td>
<td>12</td>
</tr>
<tr>
<td>12:00</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>24.54165</td>
</tr>
</tbody>
</table>

# Plenum Total Pressures

<table>
<thead>
<tr>
<th></th>
<th>1</th>
<th>2</th>
<th>3</th>
<th>4</th>
<th>5</th>
<th>6</th>
<th>7</th>
<th>8</th>
<th>9</th>
<th>10</th>
<th>11</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>2</td>
<td>3</td>
<td>4</td>
<td>5</td>
<td>6</td>
<td>7</td>
<td>8</td>
<td>9</td>
<td>10</td>
<td>11</td>
<td>12</td>
</tr>
<tr>
<td>24:00</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>24.54165</td>
</tr>
</tbody>
</table>

# Diffuser Outer Annulus Ports (11 at 6:00 and 11 at 12:00)

<table>
<thead>
<tr>
<th></th>
<th>1</th>
<th>2</th>
<th>3</th>
<th>4</th>
<th>5</th>
<th>6</th>
<th>7</th>
<th>8</th>
<th>9</th>
<th>10</th>
<th>11</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>2</td>
<td>3</td>
<td>4</td>
<td>5</td>
<td>6</td>
<td>7</td>
<td>8</td>
<td>9</td>
<td>10</td>
<td>11</td>
<td>12</td>
</tr>
<tr>
<td>12:00</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>24.54165</td>
</tr>
</tbody>
</table>

# Bottom Wall of the ECB

<table>
<thead>
<tr>
<th></th>
<th>1</th>
<th>2</th>
<th>3</th>
<th>4</th>
<th>5</th>
<th>6</th>
<th>7</th>
<th>8</th>
<th>9</th>
<th>10</th>
<th>11</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>2</td>
<td>3</td>
<td>4</td>
<td>5</td>
<td>6</td>
<td>7</td>
<td>8</td>
<td>9</td>
<td>10</td>
<td>11</td>
<td>12</td>
</tr>
<tr>
<td>12:00</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>24.54165</td>
</tr>
</tbody>
</table>

# Top Wall of the ECB

<table>
<thead>
<tr>
<th></th>
<th>1</th>
<th>2</th>
<th>3</th>
<th>4</th>
<th>5</th>
<th>6</th>
<th>7</th>
<th>8</th>
<th>9</th>
<th>10</th>
<th>11</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>2</td>
<td>3</td>
<td>4</td>
<td>5</td>
<td>6</td>
<td>7</td>
<td>8</td>
<td>9</td>
<td>10</td>
<td>11</td>
<td>12</td>
</tr>
<tr>
<td>12:00</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>24.54165</td>
</tr>
</tbody>
</table>

# ECB (Diffuser Side)

<table>
<thead>
<tr>
<th></th>
<th>1</th>
<th>2</th>
<th>3</th>
<th>4</th>
<th>5</th>
<th>6</th>
<th>7</th>
<th>8</th>
<th>9</th>
<th>10</th>
<th>11</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>2</td>
<td>3</td>
<td>4</td>
<td>5</td>
<td>6</td>
<td>7</td>
<td>8</td>
<td>9</td>
<td>10</td>
<td>11</td>
<td>12</td>
</tr>
<tr>
<td>12:00</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>24.54165</td>
</tr>
</tbody>
</table>

# Impingement Side

<table>
<thead>
<tr>
<th></th>
<th>1</th>
<th>2</th>
<th>3</th>
<th>4</th>
<th>5</th>
<th>6</th>
<th>7</th>
<th>8</th>
<th>9</th>
<th>10</th>
<th>11</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>2</td>
<td>3</td>
<td>4</td>
<td>5</td>
<td>6</td>
<td>7</td>
<td>8</td>
<td>9</td>
<td>10</td>
<td>11</td>
<td>12</td>
</tr>
<tr>
<td>12:00</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>24.54165</td>
</tr>
</tbody>
</table>
APPENDIX C: PUBLICATIONS
Conference Publications


REFERENCES

(n.d.). Retrieved January 28, 2012, from EngineerDir:

http://mysolar.cat.com/cda/layout?m=36057&x=7

http://www.hitekmfg.com/AboutUs/LandBasedGasTurbineEnginesSupport.aspx


