Experimental and Numerical Investigation of an Automotive Mixed Flow Turbocharger Turbine under Pulsating Flow Conditions

Muhammad Hasbullah Padzillah

Turbocharger Group
Department of Mechanical Engineering
Imperial College London

A thesis submitted for the degree of
Doctor of Philosophy of the Imperial College London

March 2014
Abstract

It is commonly known that the turbocharger turbine is still designed using the quasi-steady assumption despite its highly pulsating unsteady working conditions. The positioning of a turbocharger in close proximity to the exhaust valve in order to extract substantial energy ultimately necessitates a thorough investigation regarding its performance under pulsating flow conditions. This thesis presents experimental and numerical work, as well as the design of new advanced stator concept to improve turbine performance under pulsating flow conditions. A cold flow test facility is setup mainly to isolate the effect of pulsating flow conditions and therefore allowing the performance deviation from the quasi-steady approach to be properly recorded and documented. Since experimental data alone is not sufficient for understanding the detailed flow field within the turbocharger turbine stage, a complete 3-D Computational Fluid Dynamics model is developed using commercial software Ansys CFX. The model is validated against experimental data for all steady and pulsating conditions. During pulsating conditions, the incidence angle close to the rotor inlet changed significantly which directly affected the turbine performance. A study on the turbine performance improvement by aggressive reduction of nozzle vanes are conducted and experimentally tested. Results of steady and pulsating conditions suggested that the new vanes arrangement delivered significantly improved performance under both operating conditions especially at 50% speed (equivalent to 30000 rpm). At 80% speed (48000 rpm), the turbine efficiency is either similar or better (up to 8 efficiency point improvement) than the baseline arrangements.
Buat mak dan abah...
I declare this thesis to be my own work and the appropriate citations are included to acknowledge the work of others.

Muhamad Hasbullah Padzillah
‘The copyright of this thesis rests with the author and is made available under a Creative Commons Attribution Non-Commercial No Derivatives licence. Researchers are free to copy, distribute or transmit the thesis on the condition that they attribute it, that they do not use it for commercial purposes and that they do not alter, transform or build upon it. For any reuse or redistribution, researchers must make clear to others the licence terms of this work’
Acknowledgement

Alhamdulillah..

I would like to express my gratitude to my supervisor Professor Ricardo Martinez-Botas for his support both academically and emotionally. His work ethic is nothing short of inspirational. I would also like to acknowledge Universiti Teknologi Malaysia for their financial support.

Special thanks to past and present members of the Turbocharger Group, especially Srithar Rajoo, Aaron Costal, Alessandro Romagnoli, Harminder Flora, Aman Mamat, Peter Newton, Apostolos Pesiridis, Wan Saiful-Islam, Mingyang Yang, Jonathan Hey, Colin Copeland, Clemens Lorf, Lawrence Tse, Nicola Terdich, Adam Malloy, Izzal Ismail, Ibtisham Ardani, Masakazu Sakai, Uswha Khairuddin and Weilin Zhuge. These past four years would have been difficult without you.

My thanks also go to my friends, Firdaus, Zaid, Mukhlis, Hafiz, Sim, Fairuz, Zulhafiz, Badrin, Shukor, Azrudi, Nazri, Aifaa, Wong, Hazmil, Rusmizan, Shima, Nabilah, Muzammir, Shahrul, Liyana... the list goes on. You made me feel like home. To my house mates Zul and Wan, thank you for your company and encouragement, as well as the good foods.

Finally, I wish to thank my parents Hjh. Maimun and Hj. Padzillah, my brother Yusuf and my sisters Fatihah, Munirah and Syuhada, and also to the one I love who have always been supportive in all these years. This thesis is for you.
Contents

1 Introduction and literature review .................................. 30

1.1 Motivation ......................................................... 31

1.2 Energy recovery through turbocharging ......................... 34

1.2.1 History of turbocharging .................................. 34

1.2.2 Background of turbochargers ............................... 36

1.2.3 The mixed flow turbine .................................... 40

1.2.4 Experimental work on pulsating flow turbine characteristics .... 49

1.2.5 Numerical work on pulsating flow turbine characteristics .... 56

1.3 Research objectives ............................................. 70

1.4 Research scope .................................................. 70

1.5 Thesis outline .................................................... 71

1.6 Research tasks ................................................... 72
2 Experimental methodology

2.1 Introduction ....................................................... 74

2.2 Dimensional analysis and similarity approach .......................... 75

2.3 Test-Rig arrangements .............................................. 77

2.3.1 Test-Rig instrumentation ........................................ 78

2.3.1.1 Mass flow rate ............................................. 79

2.3.1.2 Pressure ..................................................... 83

2.3.1.3 Temperature ................................................. 85

2.3.1.4 Rotational speed ........................................... 86

2.3.1.5 Power measurement ....................................... 89

2.3.1.6 Miscellaneous .............................................. 91

2.4 Uncertainty analysis ................................................. 91

2.5 Data processing for unsteady measurement .......................... 93

3 Simulation methodology .............................................. 95

3.1 Introduction ....................................................... 95

3.2 Background of Computational Fluid Dynamics ........................ 95

3.3 Development of three-dimensional numerical model ................. 102

3.3.1 Geometry creation and meshing for volute, vanes and rotor ... 103

3.3.2 Model assembly ................................................. 108
3.3.3 Boundary conditions .......................... 111

3.4 Conclusion ........................................ 112

4 Steady flow turbine operation .......................... 114

4.1 Introduction ...................................... 114

4.2 Validation exercises ................................ 114

4.3 Results and discussion ............................ 116

4.4 Conclusion ...................................... 135

5 Pulsating flow turbine operation ....................... 136

5.1 Introduction ...................................... 136

5.2 Validation exercises ................................ 136

5.3 Phase shift assessment ............................ 138

5.4 Incidence angle effect on the turbine performance 145

5.5 Pulsating flow vs. steady flow analysis ............... 150

5.5.1 Flow angle comparison ......................... 150

5.5.2 Flow field comparison ........................ 155

5.6 Pulsating flow at different frequencies ............... 162

5.7 Flow ‘unsteadiness’ in the turbine stage ............... 169

5.7.1 Review on Quasi-steady Mass Flow Parameter 169

5.7.2 Lambda Parameter ............................ 171
## Contents

<table>
<thead>
<tr>
<th>Section</th>
<th>Title</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>5.8</td>
<td>Pressure distribution on the turbine blade surface</td>
<td>174</td>
</tr>
<tr>
<td>5.9</td>
<td>Conclusion</td>
<td>177</td>
</tr>
<tr>
<td>6</td>
<td>Efficiency improvement through aggressive nozzle vane reduction</td>
<td>179</td>
</tr>
<tr>
<td>6.1</td>
<td>Introduction and background theory</td>
<td>179</td>
</tr>
<tr>
<td>6.2</td>
<td>Development process</td>
<td>181</td>
</tr>
<tr>
<td>6.3</td>
<td>Review of experimental data</td>
<td>184</td>
</tr>
<tr>
<td>6.3.1</td>
<td>Steady state experimental data</td>
<td>184</td>
</tr>
<tr>
<td>6.3.2</td>
<td>Pulsating flow experimental data</td>
<td>189</td>
</tr>
<tr>
<td>6.3.2.1</td>
<td>Cycle-averaged efficiency evaluation</td>
<td>190</td>
</tr>
<tr>
<td>6.3.2.2</td>
<td>Swallowing capacity characteristics</td>
<td>196</td>
</tr>
<tr>
<td>6.3.2.3</td>
<td>Lambda parameter</td>
<td>206</td>
</tr>
<tr>
<td>6.4</td>
<td>Conclusion</td>
<td>212</td>
</tr>
<tr>
<td>7</td>
<td>Closure</td>
<td>213</td>
</tr>
<tr>
<td>7.1</td>
<td>Conclusions</td>
<td>213</td>
</tr>
<tr>
<td>7.2</td>
<td>Future work</td>
<td>218</td>
</tr>
<tr>
<td>7.2.1</td>
<td>Understanding of the turbine behaviour under pulsating flow condition</td>
<td>218</td>
</tr>
<tr>
<td>7.2.2</td>
<td>Improvement of the new nozzle vanes arrangement</td>
<td>219</td>
</tr>
<tr>
<td>References</td>
<td></td>
<td>220</td>
</tr>
<tr>
<td>Figure</td>
<td>Description</td>
<td>Page</td>
</tr>
<tr>
<td>--------</td>
<td>---------------------------------------------------------------------------------------------------</td>
<td>------</td>
</tr>
<tr>
<td>1.1</td>
<td>World CO₂ emissions by sector in 2011, [IEA 2013]</td>
<td>31</td>
</tr>
<tr>
<td>1.2</td>
<td>CO₂ emissions by sector, 1973 to 2010 [IEA 2012a]</td>
<td>32</td>
</tr>
<tr>
<td>1.3</td>
<td>CO₂ emissions from transport [IEA 2013]</td>
<td>32</td>
</tr>
<tr>
<td>1.4</td>
<td>UK domestic transport GHG emissions 2007 excluding travel across borders [Department for Transport 2009]</td>
<td>33</td>
</tr>
<tr>
<td>1.5</td>
<td>European passenger car tailpipe CO₂ trajectories (limits are shown for a typical 4-seater C-class vehicle e.g. VW Golf) [Howey et al. 2010]</td>
<td>33</td>
</tr>
<tr>
<td>1.6</td>
<td>Losses of energy of a typical light duty vehicle (%) [IEA 2012b]</td>
<td>34</td>
</tr>
<tr>
<td>1.7</td>
<td>Earliest turbocharger design by Dr. Alfred Buchi (extracted from Rajoo 2007)</td>
<td>35</td>
</tr>
<tr>
<td>1.8</td>
<td>Schematic of a turbocharged engine system, (<a href="http://www.honeywell.com">www.honeywell.com</a>)</td>
<td>37</td>
</tr>
<tr>
<td>1.9</td>
<td>3D schematic of a turbocharger, (<a href="http://www.honeywell.com">www.honeywell.com</a>)</td>
<td>37</td>
</tr>
<tr>
<td>1.10</td>
<td>Total pressure plot at the turbine inlet (Point 1)</td>
<td>39</td>
</tr>
<tr>
<td>1.11</td>
<td>(a)Effect of turbocharger to engine performance and (b) work available from ideal exhaust process, [Watson and Janota 1982]</td>
<td>39</td>
</tr>
</tbody>
</table>
List of Figures

1.13 Schematic description of a radial and a mixed-flow turbine (Rajoo and Martinez-Botas, 2008a) .................... 42
1.14 Mixed-flow blade configurations and 3D view of the rotor (Rajoo and Martinez-Botas, 2008a) ....................... 42
1.15 Efficiency of an axial, radial and mixed flow turbine against (a,b) velocity ratio (Watson and Janota, 1982) and (c) specific speed (Whitfield and Baines, 1990) .......................................................... 43
1.16 Inter-relations of mixed-flow turbine leading edge defining angles (Rajoo and Martinez-Botas, 2008a) .............. 44
1.17 Velocity ratio variation with flow inlet relative ($\beta_4$) and absolute ($\alpha_4$) angle (Rajoo and Martinez-Botas, 2008a) ............... 44
1.18 (a) Pivoting Vane VGT, Bell (1997) and (b) Sliding Nozzle Vane VGT (Holset Engineering) ......................... 47
1.19 Engine performance improvements with variable geometry turbocharger demonstrated by different researchers (Watson and Janota, 1982) ........................................... 48
1.20 Locus of instantaneous mass flow rate at one turbine entry during unsteady flow with identical (in-phase) conditions at each entry (Dale and Watson, 1986) .................................................. 51
1.21 Locus of instantaneous efficiency during unsteady flow with identical (in-phase) conditions at each entry, together with instantaneous time history of unsteady flow (Dale and Watson, 1986) .................. 52
1.22 Housing geometry of mixed-flow turbine volute: (a) old design and (b) new design (Hakeem et al., 2007) ........ 55
<table>
<thead>
<tr>
<th>Figure</th>
<th>Description</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.23</td>
<td>Schematic domain arrangements (Costall, 2007)</td>
<td>58</td>
</tr>
<tr>
<td>1.24</td>
<td>Turbine performance for predicted and experimental results (Costall, 2007)</td>
<td>58</td>
</tr>
<tr>
<td>1.25</td>
<td>Schematics of models with increasing complexity (Chiong et al., 2011)</td>
<td>60</td>
</tr>
<tr>
<td>1.26</td>
<td>CFD calculations by Barr et al. (2009) showing a smaller separation region in the backswept rotor(b); SS = Suction Surface, PS = Pressure Surface</td>
<td>62</td>
</tr>
<tr>
<td>1.27</td>
<td>Effect of the tip clearance on flow angle distribution at (a) smallest vane opening and (b) largest vane opening (Tamaki et al., 2008)</td>
<td>64</td>
</tr>
<tr>
<td>1.28</td>
<td>Inlet, exit and sliding boundary conditions (Palfreyman and Martinez-Botas, 2005)</td>
<td>66</td>
</tr>
<tr>
<td>1.29</td>
<td>Pressure at the center of the volute at given azimuth angles (a) experiment, (Karamanis, 2000) and (b) predicted (Palfreyman and Martinez-Botas, 2005)</td>
<td>66</td>
</tr>
<tr>
<td>1.30</td>
<td>Numerical domain used by Hellstrom and Fuchs, 2009</td>
<td>68</td>
</tr>
<tr>
<td>2.1</td>
<td>Schematic of turbocharger test facility at Imperial College London</td>
<td>78</td>
</tr>
<tr>
<td>2.2</td>
<td>V-cone flow meter geometry, installation, operation and maintenance manual, McCrometer, 2013</td>
<td>79</td>
</tr>
<tr>
<td>2.3</td>
<td>Flowchart of instantaneous mass flow rate measurement</td>
<td>83</td>
</tr>
<tr>
<td>2.4</td>
<td>Turbine rotational speed as the function of the 16-bit counter input (Extracted from Rajoo, 2007)</td>
<td>87</td>
</tr>
<tr>
<td>2.5</td>
<td>Main components of the eddy current dynamometer (Extracted from Szymko, 2006)</td>
<td>88</td>
</tr>
<tr>
<td>Figure</td>
<td>Description</td>
<td>Page</td>
</tr>
<tr>
<td>--------</td>
<td>------------------------------------------------------------------------------</td>
<td>------</td>
</tr>
<tr>
<td>2.6</td>
<td>Turbine rotational speed (Extracted from Szymko, 2006)</td>
<td>88</td>
</tr>
<tr>
<td>3.1</td>
<td>Volute and inlet duct geometry in Solidworks</td>
<td>103</td>
</tr>
<tr>
<td>3.2</td>
<td>The divided geometry of the volute</td>
<td>104</td>
</tr>
<tr>
<td>3.3</td>
<td>Final meshing and assembly of the volute</td>
<td>105</td>
</tr>
<tr>
<td>3.4</td>
<td>Description of the lean nozzle vane (Rajoo, 2007)</td>
<td>106</td>
</tr>
<tr>
<td>3.5</td>
<td>Comparison of normalized static pressure distribution at mid-span of the vane surfaces between CFD and experimental result with different vanes tip clearance (a) V1, (b) V2 and (c) V3</td>
<td>107</td>
</tr>
<tr>
<td>3.6</td>
<td>(a) Camberline and (b) Hub and shroud profile generated from Bezier Polynomial</td>
<td>109</td>
</tr>
<tr>
<td>3.7</td>
<td>3-D representation of the generated rotor blade profiles</td>
<td>110</td>
</tr>
<tr>
<td>3.8</td>
<td>Assembly of the domains in CFX-Pre</td>
<td>111</td>
</tr>
<tr>
<td>3.9</td>
<td>Total Pressure inlet boundary condition</td>
<td>112</td>
</tr>
<tr>
<td>4.1</td>
<td>Comparisons of Mass Flow Parameter between Experiment and CFD calculation</td>
<td>115</td>
</tr>
<tr>
<td>4.2</td>
<td>Comparison of total-to-static efficiency between Experiment and CFD calculation</td>
<td>116</td>
</tr>
<tr>
<td>4.3</td>
<td>Static pressure distribution across the volute stage (PR = 1.3, 30000 rpm)</td>
<td>117</td>
</tr>
<tr>
<td>4.4</td>
<td>Velocity distribution across the volute stage (PR = 1.3, 30000 rpm)</td>
<td>118</td>
</tr>
<tr>
<td>4.5</td>
<td>Location of data measurement at mid-span of vane and rotor inlet</td>
<td>120</td>
</tr>
</tbody>
</table>
List of Figures

4.6 Flow angle at the vane inlet and the rotor inlet ........................................... 121

4.7 Velocity components distribution across the span length of the rotor leading edge ................................................................. 122

4.8 Plot of total losses against the flow incidence angle at the rotor inlet ........ 123

4.9 Flow angle components distribution across the span length of the rotor leading edge ................................................................. 123

4.10 Locations of pressure and velocity contour in the rotor stage for (a) streamwise plane, (b) spanwise plane and (c) blade-to-blade plane ...... 125

4.11 Velocity and pressure distribution at each streamwise location of the rotor ......................................................................................... 127

4.12 Velocity and pressure distribution at each spanwise location of the rotor ................................................................. 129

4.13 Pressure loading on the blade at optimum operating condition of 30000 rpm ................................................................. 130

4.14 Velocity and Pressure distribution at 5%, 50% and 95% blade-to-blade location ................................................................. 131

4.15 Plot of the absolute flow angle for 360° circumferential location at the rotor inlet for different operating conditions ........................................ 133

4.16 Plot of the incidence flow angle for 360° circumferential location at the rotor inlet for different operating conditions ........................................ 133

4.17 Incidence angle at various spanwise positions for different operating conditions ......................................................................................... 134

5.1 Comparison of static pressure at 180° volute circumference between experiment and CFD calculation ........................................ 137
<table>
<thead>
<tr>
<th>Figure</th>
<th>Description</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>5.2</td>
<td>Comparison of rotor torque between experiment and CFD calculation</td>
<td>138</td>
</tr>
<tr>
<td>5.3</td>
<td>Isentropic and actual power plot as well as its instantaneous efficiency without phase-shifting at 50% speed for (a) 20 Hz and (b) 80 Hz</td>
<td>140</td>
</tr>
<tr>
<td>5.4</td>
<td>Isentropic power measurement locations</td>
<td>141</td>
</tr>
<tr>
<td>5.5</td>
<td>Total-to-static efficiency evaluated at different locations (a) without phase shifting, (b) phase shifted using peak power matching and (c) phase shifted using summation of bulk flow and sonic velocity</td>
<td>142</td>
</tr>
<tr>
<td>5.6</td>
<td>Instantaneous efficiency, isentropic power and actual power for different frequencies at 50% turbine speed</td>
<td>144</td>
</tr>
<tr>
<td>5.7</td>
<td>Instantaneous incidence angle at 50% speed</td>
<td>146</td>
</tr>
<tr>
<td>5.8</td>
<td>Velocity triangle representing change in incidence angle</td>
<td>147</td>
</tr>
<tr>
<td>5.9</td>
<td>Instantaneous incidence angle at 80% speed</td>
<td>148</td>
</tr>
<tr>
<td>5.10</td>
<td>Instantaneous efficiency, isentropic power and actual power for different frequencies at 80% turbine speed</td>
<td>149</td>
</tr>
<tr>
<td>5.11</td>
<td>Unsteady and steady conditions for flow field analysis</td>
<td>151</td>
</tr>
<tr>
<td>5.12</td>
<td>Flow angle distribution against the circumferential location at the vane inlet</td>
<td>152</td>
</tr>
<tr>
<td>5.13</td>
<td>Absolute flow angle plot at the rotor inlet circumference</td>
<td>153</td>
</tr>
<tr>
<td>5.14</td>
<td>Relative flow angle plot at the rotor inlet circumference</td>
<td>154</td>
</tr>
<tr>
<td>5.15</td>
<td>Area averaged incidence angle plot for both steady and pulsating conditions</td>
<td>155</td>
</tr>
<tr>
<td>5.16</td>
<td>Pressure contour at maximum pressure ratio condition at 50% speed and 20 Hz frequency</td>
<td>156</td>
</tr>
<tr>
<td>Figure</td>
<td>Description</td>
<td>Page</td>
</tr>
<tr>
<td>--------</td>
<td>------------------------------------------------------------------------------</td>
<td>------</td>
</tr>
<tr>
<td>5.17</td>
<td>Static pressure distribution on blade-to-blade planes for pt1, pt2 and pt4</td>
<td>157</td>
</tr>
<tr>
<td>5.18</td>
<td>Velocity distribution on blade-to-blade planes for pt1, pt2 and pt4</td>
<td>159</td>
</tr>
<tr>
<td>5.19</td>
<td>Static pressure distribution on blade-to-blade planes for pt1, pt3 and pt2</td>
<td>160</td>
</tr>
<tr>
<td>5.20</td>
<td>Velocity distribution on blade-to-blade planes for pt1, pt3 and pt2</td>
<td>161</td>
</tr>
<tr>
<td>5.21</td>
<td>Conditions of interest for the analysis at similar pressure level at different pulsating frequencies</td>
<td>163</td>
</tr>
<tr>
<td>5.22</td>
<td>Flow angle at vane inlet for all frequencies during the pressure increment instances</td>
<td>163</td>
</tr>
<tr>
<td>5.23</td>
<td>Absolute flow angle at rotor inlet for all frequencies during the pressure increment instances</td>
<td>164</td>
</tr>
<tr>
<td>5.24</td>
<td>Relative flow angle at rotor inlet for all frequencies during the pressure increment instances</td>
<td>165</td>
</tr>
<tr>
<td>5.25</td>
<td>Flow angle at vane inlet for all frequencies during the pressure decrement instances</td>
<td>166</td>
</tr>
<tr>
<td>5.26</td>
<td>Absolute flow angle at rotor inlet for all frequencies during the pressure decrement instances</td>
<td>167</td>
</tr>
<tr>
<td>5.27</td>
<td>Relative flow angle at rotor inlet for all frequencies during the pressure decrement instances</td>
<td>168</td>
</tr>
<tr>
<td>5.28</td>
<td>Mass flow parameter plot at (a) volute inlet and (b) rotor inlet</td>
<td>170</td>
</tr>
<tr>
<td>5.29</td>
<td>Strouhal, Pi and Lambda parameter plot against the flow frequency for both 50% and 80% turbine speed</td>
<td>173</td>
</tr>
</tbody>
</table>
5.30 Plot of normalized surface pressure of the blade at (a) 5% span, (b) 50% span and (c) 95% span ............................................. 175

6.1 The removable vanes and pivoting mechanisms .............................................. 181

6.2 Geometry of the plug .................................................................................. 182

6.3 Assembly of the plug onto the shroud wall ................................................. 182

6.4 The arrangement of a vaneless volute ........................................................... 183

6.5 The arrangement of a full vaned volute ....................................................... 183

6.6 The arrangement of a reduced vanes concept ........................................... 184

6.7 Steady state performance map of (a) Mass flow parameter and (b) Efficiency for V, VL and RV volute at 50% speed ...................... 185

6.8 The effect of nozzle vanes to the rotor inlet velocity triangle at similar incidence angle ......................................................... 186

6.9 Steady state performance map of (a) Mass flow parameter and (b) Efficiency for Vaneless, Vaned and RV volute at 80% speed ............... 188

6.10 Plot of cycle-averaged efficiency for all volute configurations against its actual power (turbine loading) at 30000 rpm and 40 Hz pulsating frequency ......................................................... 191

6.11 Plot of cycle-averaged efficiency for all volute configurations against its actual power (turbine loading) at 48000 rpm and 40 Hz pulsating frequency ......................................................... 192

6.12 Map of efficiency difference between RV and vaneless/full vaned volutes at all testing conditions for 50% design speed .................. 195
6.13 Map of efficiency difference between RV and vaneless/full vaned volutes at all testing conditions for 80% design speed ........................................ 195

6.14 Plot of mass flow parameter against pressure ratio at 50% speed and 20 Hz flow frequency for different volute configurations ........................................ 196

6.15 Plot of instantaneous mass flow parameter against pressure ratio at 50% speed and 40 Hz frequency for different volute configurations ........................................ 199

6.16 Plot of instantaneous mass flow parameter against pressure ratio at 50% speed and 60 Hz frequency for different volute configurations ........................................ 201

6.17 Plot of instantaneous swallowing capacity for RV volute arrangement at 30000 rpm and 40 Hz flow frequency under different loading conditions ........................................ 203

6.18 Plot of instantaneous swallowing capacity for RV volute arrangement at 48000 rpm and 40 Hz flow frequency under different loading conditions ........................................ 205

6.19 Plot of Lambda parameter for different volute arrangements at 30000 rpm turbine speed under low loading condition ........................................ 209

6.20 Plot of Lambda parameter for different volute arrangements at 30000 rpm turbine speed under medium loading condition ........................................ 210

6.21 Plot of Lambda parameter for different volute arrangements at 30000 rpm turbine speed under high loading condition ........................................ 210

6.22 Plot of Lambda parameter for different volute arrangements at 48000 rpm turbine speed under high loading condition ........................................ 211

A.1 V-Cone calibration document ........................................ 231
<table>
<thead>
<tr>
<th>Figure</th>
<th>Description</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>C.1</td>
<td>Instantaneous mass flow parameter at 30000 rpm, 20 Hz, high loading condition</td>
<td>238</td>
</tr>
<tr>
<td>C.2</td>
<td>Instantaneous mass flow parameter at 30000 rpm, 40 Hz, high loading condition</td>
<td>239</td>
</tr>
<tr>
<td>C.3</td>
<td>Instantaneous mass flow parameter at 30000 rpm, 60 Hz, high loading condition</td>
<td>239</td>
</tr>
<tr>
<td>C.4</td>
<td>Instantaneous mass flow parameter at 30000 rpm, 20 Hz, medium loading condition</td>
<td>240</td>
</tr>
<tr>
<td>C.5</td>
<td>Instantaneous mass flow parameter at 30000 rpm, 40 Hz, medium loading condition</td>
<td>240</td>
</tr>
<tr>
<td>C.6</td>
<td>Instantaneous mass flow parameter at 30000 rpm, 60 Hz, medium loading condition</td>
<td>241</td>
</tr>
<tr>
<td>C.7</td>
<td>Instantaneous mass flow parameter at 30000 rpm, 20 Hz, low loading condition</td>
<td>241</td>
</tr>
<tr>
<td>C.8</td>
<td>Instantaneous mass flow parameter at 30000 rpm, 40 Hz, low loading condition</td>
<td>242</td>
</tr>
<tr>
<td>C.9</td>
<td>Instantaneous mass flow parameter at 30000 rpm, 60 Hz, low loading condition</td>
<td>242</td>
</tr>
<tr>
<td>C.10</td>
<td>Instantaneous mass flow parameter at 48000 rpm, 20 Hz, high loading condition</td>
<td>243</td>
</tr>
<tr>
<td>C.11</td>
<td>Instantaneous mass flow parameter at 48000 rpm, 40 Hz, high loading condition</td>
<td>243</td>
</tr>
<tr>
<td>C.12</td>
<td>Instantaneous mass flow parameter at 48000 rpm, 60 Hz, high loading condition</td>
<td>244</td>
</tr>
<tr>
<td>Figure</td>
<td>Description</td>
<td>Page</td>
</tr>
<tr>
<td>--------</td>
<td>-------------</td>
<td>------</td>
</tr>
<tr>
<td>C.13</td>
<td>Instantaneous mass flow parameter at 48000 rpm, 20 Hz, medium loading condition</td>
<td>244</td>
</tr>
<tr>
<td>C.14</td>
<td>Instantaneous mass flow parameter at 48000 rpm, 40 Hz, medium loading condition</td>
<td>245</td>
</tr>
<tr>
<td>C.15</td>
<td>Instantaneous mass flow parameter at 48000 rpm, 60 Hz, medium loading condition</td>
<td>245</td>
</tr>
<tr>
<td>C.16</td>
<td>Instantaneous mass flow parameter at 48000 rpm, 20 Hz, low loading condition</td>
<td>246</td>
</tr>
<tr>
<td>C.17</td>
<td>Instantaneous mass flow parameter at 48000 rpm, 40 Hz, low loading condition</td>
<td>246</td>
</tr>
<tr>
<td>C.18</td>
<td>Instantaneous mass flow parameter at 48000 rpm, 60 Hz, low loading condition</td>
<td>247</td>
</tr>
</tbody>
</table>
List of Tables

<table>
<thead>
<tr>
<th>Table</th>
<th>Description</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>2.1</td>
<td>Sensitivity coefficient in the uncertainty analysis</td>
<td>92</td>
</tr>
<tr>
<td>3.1</td>
<td>Geometrical feature of Rotor A</td>
<td>108</td>
</tr>
<tr>
<td>3.2</td>
<td>Nodes distribution and type of mesh for all components</td>
<td>110</td>
</tr>
<tr>
<td>5.1</td>
<td>Cycle-averaged efficiency magnitude for all test conditions</td>
<td>148</td>
</tr>
<tr>
<td>5.2</td>
<td>Unsteady and steady conditions for flow field analysis</td>
<td>151</td>
</tr>
<tr>
<td>6.1</td>
<td>Stator gap distance for pulsating flow testing</td>
<td>190</td>
</tr>
<tr>
<td>6.2</td>
<td>Cycle-averaged efficiency magnitude for all test conditions at 30000 rpm turbine speed</td>
<td>193</td>
</tr>
<tr>
<td>6.3</td>
<td>Cycle-averaged efficiency magnitude for all test conditions at 48000 rpm turbine speed</td>
<td>193</td>
</tr>
<tr>
<td>6.4</td>
<td>Efficiency difference between RV volute with vaneless and full vaned volute arrangements (%) at 30000 rpm turbine speed</td>
<td>193</td>
</tr>
<tr>
<td>6.5</td>
<td>Efficiency difference between RV volute with vaneless and full vaned volute arrangements (%) at 48000 rpm turbine speed</td>
<td>194</td>
</tr>
<tr>
<td>6.6</td>
<td>Values of Strouhal, II and Λ for 50% turbine speed</td>
<td>207</td>
</tr>
</tbody>
</table>
List of Tables

6.7 Values of Strouhal, Π and Λ for 80% turbine speed . . . . . . . . . . 208

B.1 Rotor A defining characteristics . . . . . . . . . . . . . . . . . . . . 233
## Nomenclature

**Acronyms**

<table>
<thead>
<tr>
<th>Acronym</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>ACT</td>
<td>active control turbocharger</td>
</tr>
<tr>
<td>AIAA</td>
<td>American institute of aeronautics and astronautics</td>
</tr>
<tr>
<td>ASME</td>
<td>American society of mechanical engineer</td>
</tr>
<tr>
<td>BS</td>
<td>British standard</td>
</tr>
<tr>
<td>CFD</td>
<td>computational fluid dynamics</td>
</tr>
<tr>
<td>CHP</td>
<td>combined heat and power</td>
</tr>
<tr>
<td>CTA</td>
<td>constant temperature anemometry</td>
</tr>
<tr>
<td>DNS</td>
<td>direct numerical simulation</td>
</tr>
<tr>
<td>FGT</td>
<td>fixed geometry turbine</td>
</tr>
<tr>
<td>FIR</td>
<td>finite impulse response</td>
</tr>
<tr>
<td>FP</td>
<td>field point</td>
</tr>
<tr>
<td>GHG</td>
<td>green house gas</td>
</tr>
<tr>
<td>HPC</td>
<td>high performance computer</td>
</tr>
<tr>
<td>ICE</td>
<td>internal combustion engine</td>
</tr>
<tr>
<td>IEA</td>
<td>international energy agency</td>
</tr>
<tr>
<td>LDV</td>
<td>laser doppler velocimetry</td>
</tr>
</tbody>
</table>
Nomenclature

NACA  national advisory committee for aeronautics
NA    naturally aspirated engine
NI    National instruments
PCI   peripheral component interconnect
PS    pressure surface
RANS  Reynolds-Averaged Navier-Stokes
RV    Reduced vanes
SS    suction surface
VAT   variable area turbocharger
VGT   variable geometry turbine
VNT   variable nozzle turbocharger

Greek Symbols

\( \alpha \)   angular acceleration
\( \alpha \)   absolute flow angle
\( \beta \)   relative flow angle
\( \varepsilon \) turbulent dissipation rate
\( \eta \)   efficiency
\( \gamma \) specific heat ratio
\( \Lambda \) lambda parameter, \( \pi \cdot St \)
\( \mu \)   dynamic viscosity
\( \nabla \) vector differential operator
\( \nu \)   kinematic viscosity
\( \nu_T \) turbulent viscosity
### Nomenclature

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\omega$</td>
<td>angular speed</td>
</tr>
<tr>
<td>$\Pi$</td>
<td>pi parameter, $\frac{2\Delta \rho}{\rho_0}$</td>
</tr>
<tr>
<td>$\rho$</td>
<td>density</td>
</tr>
<tr>
<td>$\tau$</td>
<td>torque</td>
</tr>
<tr>
<td>$\tau_\omega$</td>
<td>wall shear stress</td>
</tr>
<tr>
<td>$\theta$</td>
<td>angle</td>
</tr>
</tbody>
</table>

### Roman Symbols

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>$A$</td>
<td>area</td>
</tr>
<tr>
<td>$C$</td>
<td>turbine absolute velocity</td>
</tr>
<tr>
<td>$C_d$</td>
<td>discharge coefficient</td>
</tr>
<tr>
<td>$C_m$</td>
<td>meridional velocity</td>
</tr>
<tr>
<td>$C_\theta$</td>
<td>flow frequency</td>
</tr>
<tr>
<td>$D$</td>
<td>diameter</td>
</tr>
<tr>
<td>$d$</td>
<td>inside diameter</td>
</tr>
<tr>
<td>$E$</td>
<td>hotwire voltage</td>
</tr>
<tr>
<td>$f$</td>
<td>flow frequency</td>
</tr>
<tr>
<td>$F_a$</td>
<td>material thermal expansion factor</td>
</tr>
<tr>
<td>$FFT$</td>
<td>fast Fourier transform</td>
</tr>
<tr>
<td>$FS$</td>
<td>full scale</td>
</tr>
<tr>
<td>$I$</td>
<td>polar moment of inertia</td>
</tr>
<tr>
<td>$k_1$</td>
<td>flow coefficient</td>
</tr>
<tr>
<td>$k$</td>
<td>turbulent kinetic energy</td>
</tr>
<tr>
<td>$K$</td>
<td>von Karman constant</td>
</tr>
<tr>
<td>Symbol</td>
<td>Definition</td>
</tr>
<tr>
<td>--------</td>
<td>----------------------------</td>
</tr>
<tr>
<td>$L_0$</td>
<td>domain length</td>
</tr>
<tr>
<td>$M$</td>
<td>Mach number</td>
</tr>
<tr>
<td>$\dot{m}$</td>
<td>mass flow rate</td>
</tr>
<tr>
<td>$MFP$</td>
<td>mass flow parameter</td>
</tr>
<tr>
<td>$N$</td>
<td>rotational speed</td>
</tr>
<tr>
<td>$Nu$</td>
<td>Nusselt number</td>
</tr>
<tr>
<td>$P$</td>
<td>pressure</td>
</tr>
<tr>
<td>$PR$</td>
<td>pressure ratio</td>
</tr>
<tr>
<td>$Pr$</td>
<td>Prandtl number</td>
</tr>
<tr>
<td>$R$</td>
<td>gas constant</td>
</tr>
<tr>
<td>$Re$</td>
<td>Reynolds number</td>
</tr>
<tr>
<td>$RSS$</td>
<td>root-sum-square</td>
</tr>
<tr>
<td>$St.*$</td>
<td>normalized strouhal number</td>
</tr>
<tr>
<td>$St.(p)*$</td>
<td>pressure modified strouhal number</td>
</tr>
<tr>
<td>$T$</td>
<td>temperature</td>
</tr>
<tr>
<td>$t$</td>
<td>time</td>
</tr>
<tr>
<td>$U$</td>
<td>turbine tip speed</td>
</tr>
<tr>
<td>$U_1$</td>
<td>unit conversion constant</td>
</tr>
<tr>
<td>$\langle U \rangle$</td>
<td>mean flow velocity</td>
</tr>
<tr>
<td>$u^+$</td>
<td>non-dimensionalized velocity</td>
</tr>
<tr>
<td>$u_\tau$</td>
<td>friction velocity, $\left( \frac{\tau}{\rho} \right)^{\frac{1}{2}}$</td>
</tr>
<tr>
<td>$U$</td>
<td>flow velocity</td>
</tr>
<tr>
<td>$V_0$</td>
<td>domain volume</td>
</tr>
</tbody>
</table>
Nomenclature

$VR$ velocity ratio

$\dot{W}$ power

$x$ location along streamwise direction

$Y$ gas expansion factor

$y^+$ non-dimensionalized distance from wall

**Subscripts**

0 total condition

1 inlet condition

5 exit condition

$act$ actual condition

actual actual condition

calib calibration condition

cyc$_{avg}$ cycle-averaged parameter

eqv equivalent parameter

fluc fluctuation parameter

inst instantaneous parameter

is isentropic

mean mean parameter

meas measured parameter

meas$_{plane}$ measurement plane condition

s static condition

test-rig test rig condition

$ts$ total-to-static
<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>$v$</td>
<td>full vaned volute</td>
</tr>
<tr>
<td>$vl$</td>
<td>vaneless volute</td>
</tr>
</tbody>
</table>
Chapter 1

Introduction and literature review

The ever-increasing demand for low carbon applications in the automotive industry has intensified the development of highly efficient engines and energy recovery devices. Even though there have been significant developments in alternative powertrains such as hybrid and electric vehicles, their full deployment is hindered by high costs and an unattractive life-cycle energy and emissions balance. Thus powertrains based on highly efficient internal combustion engines are still considered to be the mainstream solution for years to come. Traditionally, the turbocharger has been an essential tool to boost the engine power especially the diesel engine. However, in recent years it is seen as an enabling technology for engine downsizing of all internal combustion engines. In reality, a turbocharger turbine coupled to an internal combustion engine operates in a highly pulsating exhaust flow. This is due to the nature of a reciprocating engine that exhales the exhaust gas at a specific frequency. Heywood (1988) indicated that it is necessary that the turbine should be designed carefully in order to achieve sufficient efficiency in such conditions. There are numerous studies looking into the complex interaction of the pulsating exhaust gas with the turbocharger turbine, however the phenomena is still not fully integrated into the design process. While the conventional practice in the industry is still to design and match the turbine to an engine based on steady performance maps, there have been significant developments and research on integrating the knowledge of pulsating flow turbine characteristics into the turbine design.
Chapter 1. Introduction and literature review

1.1 Motivation

The broader scope of this project is closely linked to the global strategy to actively reduce carbon emissions to the atmosphere, particularly from the transportation sector. According to the International Energy Agency (IEA) in 2013, transport alone was responsible for 22% of global energy-related CO₂ emissions (see Figure 1.1). This proportion could become even greater as other sectors are decarbonised (see Figure 1.2). From the entire transport sector, road transportation alone contributed more than 72% of the total emission, thus signifying the need for actions. As for the UK government, they have committed to an ambitious target of reducing greenhouse gas (GHG) emission by 50% by 2027 and 80% by 2050 from 1990 levels (IEA, 2012a). In order to achieve this target, aggressive actions from the policy makers, and not to mention the technology developers, are necessary. Figure 1.4 shows the UK domestic transport GHG emission contributors from all types of vehicles in 2007. It can be clearly seen that passenger cars and heavy goods vehicles dominated and contributed almost 80% of the total GHG emission from the transport sector in the UK.

![Figure 1.1: World CO₂ emissions by sector in 2011, (IEA, 2013)](image)

Figure 1.1 shows the projections of CO₂ emissions of the European passenger car fleet until 2050. In order to achieve these trajectories, more and more stringent emissions regulations have been imposed over the years thus forcing the industry to move towards the development of highly efficient state-of-the-art engine technology. One of the key enablers for low emission internal combustion engines is turbocharg-
Figure 1.2: CO₂ emissions by sector, 1973 to 2010 (IEA, 2012a)

Figure 1.3: CO₂ emissions from transport (IEA, 2013)
Figure 1.4: UK domestic transport GHG emissions 2007 excluding travel across borders (Department for Transport, 2009)

Figure 1.5: European passenger car tailpipe CO₂ trajectories (limits are shown for a typical 4-seater C-class vehicle e.g. VW Golf) (Howey et al., 2010)

1.1. Motivation
ing technology. Turbocharging directly results in the increase of the power density of an engine. Therefore, it is possible to downsize the turbocharged engine while maintaining its power output. One of the major benefits of a downsized engine is the smaller contact surfaces between moving parts and as such results in the friction reduction. This inevitably results in the efficiency increment of an engine.

![Figure 1.6: Losses of energy of a typical light duty vehicle (%) (IEA, 2012b)](image-url)

1.2 Energy recovery through turbocharging

1.2.1 History of turbocharging

Turbocharging from a historical perspective goes back to the late 1800s, when the German inventor Gottlieb Daimler and Rudolf Diesel were designing the mechanical supercharger as a way to boost the performance of engines. The unavoidable energy losses in the internal combustion engine are due to friction and heat sealing losses, and a major contributor is the heat energy in the exhaust gas, hence the needs for an energy recovery system that can restore this wasted energy to useful work. In November 1905, a Swiss engineer, Dr. Alfred Buchi received patent No. 204630 from The Imperial Patent Office of the German Reich for a “combustion machine consisting of a compressor (turbine compressor), a piston engine, and a turbine in sequential arrangement” (see Figure 1.7).

The initial stage of turbocharger development was focused on large engine applications such as for marine and aircraft engines, as well as for trucks. The first
Chapter 1. Introduction and literature review

Figure 1.7: Earliest turbocharger design by Dr. Alfred Buchi (extracted from Rajoo (2007))

turbocharged truck engine was built by the “Swiss Machine Works Saurer” in 1938. For passenger car applications, turbocharged engines entered the market in 1962 but quickly disappeared due to reliability problems. Conversely, in the late 1980s the increasingly stringent emission regulations greatly enhanced the production of truck engines equipped with turbochargers, and today the vast majority of the truck engines are turbocharged. Turbocharged engines found their way into motorsport in the 1970s and started to become popular for passenger cars. However, they disappeared again from the public domain due to its cost and the issue of ‘turbo-lag’\(^1\) that was not acceptable by the drivers. These problems have since been addressed by appropriate turbocharger matching, selection of the high temperature materials, and utilization of advanced instrumentations to improve the transient response of the machine.

In 1987, the introduction of a turbocharged diesel passenger car, the Mercedes-Benz SD300, followed by the VW Golf Turbodiesel a few years later, marked the major breakthrough in turbocharger applications to the passenger car. It is now widely accepted as a means of increasing engine efficiency and reducing emissions now rather than to simply increase engine performance in order to achieve compliance with progressively more stringent emissions regulations (BorgWarner Turbo

\(^1\)the delayed response of turbocharger due to finite time taken for the gas to fill the volume of the manifold system
1.2.2 Background of turbochargers

A naturally aspirated (NA) internal combustion engine (ICE) can be viewed as a breathing mechanism that inhales the ambient air and exhales the combustion products. In a 4-stroke ICE, the inhaled ambient air is mixed with the fuel at a certain air-fuel ratio, compressed and burned in the combustion cylinder that translates into reciprocal movement of the piston which in turns rotates the crankshaft. The power output from the crankshaft is then delivered to the wheels that result in vehicle movement. The combustion cylinder then exhales the combustion product via exhaust valves back to ambient.

However, when the same NA engine operates at an elevated altitude, although the same volume of air is drawn into the combustion chamber during the intake stroke, the reduction of air density would result in a reduced mass of air. As the combustion power output is the function of the mass of the air (the number of air molecules and not the volume), the power output to the crankshaft would ultimately decrease. This effectively reduces the engine power output and its driveability especially as the altitude increases (for example, climbing hills). Therefore, it would be beneficial if the air is compressed before being fed into the combustion chamber, the purpose for which the turbochargers and superchargers are designed. The following discussions are focused on the turbocharger since the turbocharger turbine is the focus of this research.

A turbocharger is a device that is capable of recovering the energy available in the hot exhaust gas by means of a rotating turbine rotor. This rotor is connected to the compressor via a common shaft. Therefore, the energy available in the turbine can be transferred directly to the compressor in order to compress the intake air which is then fed into the combustion chamber (Japikse and Baines 1994, Pulkrabek 2007). A schematic of a turbocharged engine is illustrated in Figure 1.8.

A turbocharger consists of several components, namely the turbine volute, com-
Chapter 1. Introduction and literature review

Figure 1.8: Schematic of a turbocharged engine system, (www.honeywell.com)

Figure 1.9: 3D schematic of a turbocharger, (www.honeywell.com)

1.2. Energy recovery through turbocharging
pressor scroll, turbine wheel, compressor wheel, and a central shaft as is shown in Figure 1.9. During its operation, the hot exhaust gas that originated from the combustion of the air-fuel mixture inside the combustion chamber in each cylinder is directed into the exhaust manifold before entering the turbine volute. Within the volute stage, the hot gas is forced to turn and follow the volute shape, resulting in the development of large tangential velocity. This large tangential velocity gas, also with high temperature and pressure in turn rotates the turbine wheel. Furthermore, the reduction of cross-sectional area of the volute as the flow propagate downstream its circumference further forces the fluid to flow into the rotating turbine rotor passage. Downstream of the turbine exit, the fluid is discharged to ambient air.

The rotation of the turbine wheel therefore rotates the compressor rotor, allowing ambient air drawn into the turbocharger to be compressed before entering the engine intake duct. The high density and high temperature air due to compressive heating (Pulkrabek, 2007) is often channelled through a heat exchanger to reduce its temperature and further increase its density before entering the combustion chamber. The air is then mixed with fuel and burnt during the combustion process. Afterwards, the hot combustion products are released into the exhaust manifold where it will enter the turbocharger turbine and the process repeats continuously.

At a constant engine speed, the combustion process inside the engine cylinder occurs at a certain frequency due to the nature of a reciprocating device. This behaviour translates into the fact that the discharge of the combustion products through the exhaust also takes place at a certain frequency. Therefore, the flow within the exhaust manifold is always pulsating and ultimately introduces a significant unsteadiness at the turbine inlet as indicated Figure 1.10.

From Figure 1.10, it can be seen that the flow entering the turbocharger turbine wheel is not steady but always pulsating. However, due to the lack of understanding of the turbine behaviour under pulsating flow conditions, the turbine wheel at the moment is still matched to an engine based on steady flow performance map. The reciprocating nature of the ICE results in coupling an unsteady flow to a steady flow device (the turbine) and could be seen as an ‘unhappy marriage’, and a challenging
engine-turbocharger matching procedure. At the moment, the unsteady phenomena within the turbocharger turbine are still not fully characterized and understood due to the intricate nature of the flow field as well as the complex turbocharger geometry. This presents an opportunity to further improve the engine-turbocharger matching by understanding the turbine interaction with the incoming pulsating flow. From the fundamental thermodynamics point of view, the advantage of extra power available by the turbocharged engine can be seen by understanding its effect on the ideal Dual Cycle of IC Engine plotted in Figure 1.11 (Watson and Janota, 1982).

An idealized thermodynamic cycle in a naturally aspirated engine and turbocharged engine is shown in Figure 1.11 (a) and the energy available during the
Chapter 1. Introduction and literature review

exhaust process is shown in Figure 1.11(b). In Figure 1.11(a), process 1-2 represent
compression stroke, 2-3 is the combustion process followed by combustion process at
constant pressure 3-4. 4-5 represent the power stroke where the combustion cham-
ber expands and 5-1 is the exhaust stroke where the exhaust gas is removed from
combustion chamber thus lowering the pressure back to ambient pressure. In Figure
1.11(b), the opening of exhaust valve is marked by position 1, therefore the isen-
tropic energy available in the exhaust gas is encapsulated in 1-2-3 region in Figure
1.11(b). The effect of turbocharger is to increase the pressure inside combustion
chamber thus increasing the encapsulated area of thermodynamic cycle 1-2-3-4-5 in
Figure 1.11(a). The net work output is represented as the integral of the pressure
with volume as shown in Equation 1.1

\[ W = \int PdV \]  

(1.1)

Also, in order to accommodate such high flowrates of the exhaust gas, a turbine
wheel is designed to operate at the high rotational speed (usually in access of 100
000 rpm). At particularly high speed engine operation, the rotational velocity of the
turbine and compressor wheel could be dangerously high and therefore could over
compress the air. To avoid this from happening, a wastegate system (see Figure
1.12) is installed to allow the exhaust to bypass the turbine wheel and therefore
limiting the boost pressure.

1.2.3 The mixed flow turbine

As a mixed flow turbine is the integral part of this research, it is felt necessary
to perform a review on the particular device. A comprehensive review on mixed
flow turbines is already made available by Rajoo and Martinez-Botas (2008a) and
therefore only the imperative aspects of the mixed flow turbine is discussed in this
section.

One of the limitations associated with the radial turbine is that its geometrical
configuration requires the inlet blade angle to be 0° in order to maintain the ‘radial fibre requirement’. This prevents too great bending stress exerted on the turbine blade due to the high centrifugal force. This condition forces the peak efficiency of the radial turbine to be fixed at the particular velocity ratio of 0.707 and prohibits the potential for reducing the optimum velocity ratio (increasing pressure ratio) for better operation under pulsating flow condition. The invention of the mixed flow turbine seems to have brought said potential by allowing the blade to have a non-zero inlet blade angle while maintaining the radial fibre requirement.

The underhood space limitation for an automotive turbocharger results in installation close to the exhaust valve, therefore exposing the turbine wheel to a highly pulsating flow. It is therefore sensible to adjust the turbine geometry to extract the energy available at high pressure ratio instance of the incoming pulsating flow, rather than designing turbine geometry for cycle-averaged conditions. Meanwhile, the pressure ratio is inversely related to velocity ratio as indicated in Equation 1.2.

---

1. It is common for the turbine designer to design blade that extends radially outward at any axial location
2. In an ideal analysis, peak efficiency can be shown to occur at \( VR = \frac{1}{\sqrt{2}} \approx 0.7 \)
\[ \frac{U}{C_{is}} = \frac{U}{\sqrt{2c_p T_{0,inlet} \left[ 1 - \left( \frac{P_{s,exit}}{P_{0,inlet}} \right)^{\frac{2 - 1}{\gamma}} \right]}} \] (1.2)

Figure 1.13: Schematic description of a radial and a mixed-flow turbine (Rajoo and Martinez-Botas, 2008a)

Figure 1.14: Mixed-flow blade configurations and 3D view of the rotor (Rajoo and Martinez-Botas, 2008a)

Mixed flow turbine geometry is achieved by radially sweeping the leading edge of a radial turbine as shown in Figure 1.13. This modification results in reduced flow path curvature and reduced formation of secondary flow as compared to its
radial counterpart (Palfreyman and Martinez-Botas, 2002). Figure 1.14 shows the geometrical characteristic of a mixed flow turbine which is defined by the blade angle, cone angle and camber angle. The cone angle is effectively what differentiates the mixed flow turbine and the radial turbine (where it is fixed at 90°). By altering the cone angle, it is then possible for the blade angle to have a non-zero value thus allowing the possibility to shift the peak efficiency point to a lower velocity ratio magnitude, as represented in Figure 1.15. The relationship between the blade angle, cone angle and camber angle is obtained from Equation 1.3 as indicated by Whitfield and Baines (1990).

\[
\tan \beta_B = \cos \lambda \tan \phi
\]  

(1.3)

Figure 1.15: Efficiency of an axial, radial and mixed flow turbine against (a,b) velocity ratio (Watson and Janota, 1982) and (c) specific speed (Whitfield and Baines, 1990).

The relationship between the mixed flow turbine defining angles are illustrated in Figure 1.16. The representation of the effect of the inlet blade angle to the velocity ratio is shown in Figure 1.17. It can be seen from Figure 1.17 that the application of different blade angle have a significant impact on velocity ratio as described earlier.
Chapter 1. Introduction and literature review

1.2. Energy recovery through turbocharging

Figure 1.16: Inter-relations of mixed-flow turbine leading edge defining angles (Rajoo and Martinez-Botas, 2008a)

Figure 1.17: Velocity ratio variation with flow inlet relative ($\beta_4$) and absolute ($\alpha_4$) angle (Rajoo and Martinez-Botas, 2008a)
The design methodology of the mixed flow turbine was developed by a number of researchers. Early research by Wallace and Blair (1965) aimed at improving the swallowing capacity of the turbocharger turbine by using the mixed flow turbine. Baines et al. (1979) tested two mixed flow turbines and achieved similar agreement of turbine performance to that of Wallace and Blair (1965). Furthermore, Abidat (1991) and Abidat et al. (1992) developed a new method of generating the blade profile by utilizing Bezier polynomials. They were able to achieve a similar test result to Wallace and Blair (1965) and Baines et al. (1979). Moreover, a systematic approach to optimize design of mixed flow turbines was explained in great detail by Chen and Baines (1992) who attempted to minimize energy losses by reducing the inlet and exit velocities of the rotor. Chen and Baines (1992) later concluded that the turbine design can be optimized by setting the exit swirl angle to zero or some positive value, thereby balancing reduced internal loss and increased exit loss.

The ability of a mixed flow turbine, compared to its radial counterpart, to increase swallowing capacity enables the use of a smaller turbine rotor for a given mass flow target. This consequently reduces turbine inertia, which leads to better turbocharger response – a very desirable characteristic. The potential improvements that the mixed flow turbine offers have been a great motivation for much research into the application of mixed flow turbines to ICEs, such as Yamaguchi et al. (1984), Chou and Gibbs (1989), Naguib (1986), and Minegashi et al. (1995).

In the attempt to improve the effectiveness of the turbocharger turbine, the application of nozzle vanes was introduced to provide better flow guidance into the turbine wheel. Although the precise reason for the enhancement of turbine efficiency is still not absolutely clear, the addition of nozzle vanes were widely adopted, especially for heavy duty vehicles. While a Fixed Geometry Turbine (FGT) nozzle offers performance improvement only at very specific operating conditions, a Variable Geometry Turbine (VGT) that is capable of altering the flow area to suit particular operating conditions is also used, but at the expense of reliability and increased product cost.

The design intent of a VGT is to enhance the efficient operating range of tur-
bocharger turbine, enable better transient response and maintain sufficient boosting pressure at the intake manifold for a wide range of engine speed. Conventional FGTs are limited by the workable range of turbine due to choking when the exhaust flow become too large (high engine speed), and of the compressor due to stalling when the exhaust flow become too small (low engine speed). This behaviour effectively means that during high engine speed, the turbine rotor is not able to discharge all the exhaust gas, causing pressure accumulation upstream of the rotor (a phenomena called “back pressure”). On the other hand, at low engine speed, the developed turbine power is insufficient for the compressor to provide the desired boost pressure.

A VGT operates by increasing the flow area during high speed engine operation, to prevent over-boosting, and decreasing the flow area during low speed operation in order to increase the fluid velocity and momentum impacting the turbine rotor. Apart from that, the application of VGT will allow the wastegate system to be removed due to the ability of VGT control flow into the rotor thus preventing over-boost pressure. However, the major drawback in the application of VGT is that its actuating mechanism would significantly increase the number of components and therefore introduce the reliability issue. Two types of VGT mechanisms are shown in Figure 1.18.

The history of VGT goes back over 30 years when Watson and Janota (1982) suggested that VGT could be a potential solution for engines operating in highly transient conditions. Nevertheless, its development continues to be restricted by durability and reliability concerns thus limiting the applicability of VGT for commercial uses. Figure 1.19 shows the improved brake power, specific fuel consumption and reduced emissions of VGT as compared to a fixed geometry unit.

Wallace et al. (1981) mentioned that the VGT is the next stage of turbocharging technology at that time, and conducted a series of engine tests equipped with VGT and scrutinized the improvement achieved. Furthermore, it was found that the engine transient response improved but was accompanied with a loss of turbine efficiency, suggesting the need for an optimal control strategy to maximize on-engine improvement.
A few methods of controlling the flow in the variable geometry turbocharger have been suggested. To date, the method of moving wall/sliding nozzle and the variable nozzle area through pivoting mechanism are widely used and the most reliable (see Figure 1.18) as compared to the other suggested designs. For VGTs with a pivoting vane mechanism, the nozzles are connected on a common ring that regulates the flow area by rotating the nozzles. As for the sliding nozzle mechanism, the nozzles maintain a fixed angle but the flow area upstream of the nozzle leading edge is controlled by the sliding ring. Both mechanisms require various engine parameters to be taken into account to achieve the optimum compromise between fast torque
Figure 1.19: Engine performance improvements with variable geometry turbocharger demonstrated by different researchers (Watson and Janota, 1982) response, fuel economy, low emissions and engine safety (Moody, 1986).

Design of the VGT has greatly revolutionized since the development of mixed flow turbine. Since the turbine inlet was radially swept at certain angle (cone angle), it was possible to use design a non-zero blade angle at the leading edge while maintaining radial fibre. This is something that cannot be achieved with the conventional radial turbine and has allowed more degree of freedom for turbine designers. However, the mixed flow turbine imposes significant challenges in finding a suitable vane to match the geometrical properties of its rotor leading edge. Wallace and Pasha (1972) pointed out that an ‘ideal’ nozzle vane ring specifically designed for mixed flow rotor inlet is essential in order to obtain maximum performance of VGT. For instance, the use of conventional straight vanes will create an uneven interspace region between the vane trailing edge and the rotor leading edge, resulting in non-uniformity of the flow in spanwise direction (Rajoo and Martinez-Botas, 2008a).

Nevertheless, Baets et al. (1998) utilized a set of straight pivoting nozzle vanes to control the inlet area to a mixed flow turbine, arranged in such a way that the clearance leakage is minimized. They indicated that for the specific design of
the pivoting vanes, reduction of efficiency was mostly due to the sealing loss and nozzle ring geometry loss. However, they recorded successful engine part-load fuel consumption and exhaust smoke test results.

Recent development of the VGT includes the experimentally-tested lean vanes designed by Rajoo and Martinez-Botas (2006), which were designed to match the 3D geometry of the mixed flow turbine at the leading edge. Indeed, the design and experimental data obtained by Rajoo (2007) is used for validation throughout the current research. Another quite recent invention of an advanced type of VGT is the ‘Active Control Turbocharger’ (ACT) by Pesiridis (2007) and Pesiridis and Martinez-Botas (2005, 2006, 2007). The sliding vane actuating mechanism was modified to adapt to the pulsating flow nature of the exhaust gas by artificially inducing sinusoidal motion. Therefore, the energy contained in the pulsating flow is extracted on a pulse-to-pulse basis by “actively” controlling the inlet flow area.

1.2.4 Experimental work on pulsating flow turbine characteristics

The earliest experimental study on the effect of unsteady flow on turbine performance was reported by Wallace and Blair (1965). They systematically attempted to evaluate the influence of several parameters such as the pulse frequency, pulse form, pulse amplitude, pipe length, pipe diameter and turbine speed on the performance of the radial turbine. They also indicated that a multiple entry turbine effectively operates under partial admission conditions during pulsating flow operation. Wallace and Blair (1965) later concluded that the quasi-steady flow hypothesis became progressively weaker as pulse frequency increased. Furthermore, they also observed that short duration pulses gave higher power outputs than long duration pulses, and that a smaller pipe area also provided higher turbine power output. As for the partial-admission theory, results from this work indicated that the use of full-admission efficiencies lead to a relatively small overestimation of power, while using the partial-admission efficiencies resulted in a drastic underestimation.

1.2. Energy recovery through turbocharging
Benson and Scrimshaw (1965) conducted an experimental study to investigate the effect of flow frequency and velocity on the turbine performance. They also attempted to associate partial and full admission steady flow turbine performance data to the unsteady analysis. Despite the lack of instantaneous measurements (except for the static pressure), Benson and Scrimshaw (1965) observed significant underestimation of the pulse-averaged mass flow rate and power output when the steady state partial admission test data were used, as previously observed by Wallace and Blair (1965). Furthermore, Benson and Scrimshaw (1965) also indicated that the pulse-averaged turbine efficiency was greater under pulsating flow than that under steady flow conditions, their magnitude depending on pulse frequency and turbine speed.

Pischinger and Wunshe (1977) developed a test rig specifically to obtain both full and partial admission turbocharger turbine maps. The evaluation of these maps became one of the most important research works in the past, partly due to the inability to obtain accurate instantaneous flow measurements in order to understand the turbine characteristics under pulsating flow operation. They later found that there is significant deviation between the turbine performance under full and partial admission.

Almost a decade later Dale and Watson (1986) developed and tested a small, high-speed turbocharger twin-entry turbine under both steady and pulsating flow conditions. They also utilized a 12 kW eddy current dynamometer instead of a compressor as the loading device, thus opening up the possibility for map extension. In this work, apart from the instantaneous pressure measurement, for the first time the instantaneous mass flow is reported - measured using hot-wire anemometry. Instantaneous torque was measured by means of a load cell and the turbine inertia, as well as the precise measurement of the turbine speed in order to calculate its acceleration and deceleration. As for the steady flow partial admission testing, they found out that the maximum efficiency did not necessarily occur at equal inlet conditions, even with a symmetrical flow housing. They also found that the lowest efficiency occurs when there was absolutely no flow through one entry. Furthermore, Dale and Watson

1 The use of compressor limits the turbine operational range due to surge and stall
[1986] also found that for constant pressure ratio and turbine speed, the instantaneous mass flow rate departed from that measured under steady flow. Although the mean mass flow was very close to the steady flow line, the mean efficiency was lower compared to the corresponding steady flow efficiency. Instantaneous measurements revealed for the first time that non-dimensional mass flow rate and efficiency formed hysteresis loops that encapsulated a portion of the quasi-steady assumption in the map, as shown in Figure 1.20 and Figure 1.21 respectively.

![Figure 1.20: Locus of instantaneous mass flow rate at one turbine entry during unsteady flow with identical (in-phase) conditions at each entry (Dale and Watson, 1986)](image)

Three years later [Capobianco and Gambarotta (1989)] conducted a study to investigate the departure of pulsating flow conditions from the quasi steady counterpart. In this study the only instantaneous parameters measured were the upstream and downstream pressure. [Capobianco and Gambarotta (1989)] claimed that the temperature and mass flow rate have a low response characteristic and therefore justified the use of their mean values. They later indicated that the averaged turbine swallowing capacity and power under pulsating flow conditions were always higher than those found during steady flow conditions. However, they did not find any link between the unsteady characteristics and the flow frequency. Furthermore, this work also addressed the need for steady state curve extrapolation in order to
Figure 1.21: Locus of instantaneous efficiency during unsteady flow with identical (in-phase) conditions at each entry, together with instantaneous time history of unsteady flow [Dale and Watson, 1986] match the operating range under pulsating flow conditions.

Capobianco and Gambarotta (1992) continued their work on pulsating flow turbine characteristics by using a Variable Area Turbocharger (VAT) and a Variable Nozzle Turbocharger (VNT). The geometry of the VAT in this study was varied by movement of the tongue in order to adjust the volute inlet area. On the other hand, the geometry of the VNT was varied using its pivoting nozzle. A fixed geometry turbine was also included in the study but the benefits of the variable geometry systems were apparent. Capobianco and Gambarotta (1992) indicated that the pulse shape at the turbine inlet was affected by the frequency as well as the variable geometry system. This is due to the observation where the presence of moving vanes seemed to have introduced significant oscillations in the pressure profile at the outlet. As in the previous work, an appropriate link between turbine performance and the pulsating frequency and amplitude still could not be found. Furthermore, the outcome of this work suggested that the use of proper control strategies were necessary in order to take full advantage of the variable geometry system.

1.2. Energy recovery through turbocharging
Baines et al. (1994) conducted a systematic study on the effect of pulse flow inlet conditions on the performance of a nozzleless radial inflow turbine. This work indicated that for out-of-phase pulses and low frequency conditions, there exists some reverse flow from one inlet to the other. As the frequency increases, the turbine performance indicated a closer relationship with the steady state conditions although minor differences still existed. Baines et al. (1994) also developed a “filling-and-emptying” to model the volume of the volute, where mass can transiently accumulate. Subsequently, the flow was channelled to the turbine wheel, modelled using the quasi-steady assumption. By using this model, Baines et al. (1994) managed to obtain good agreement with in-phase measurement over a range of operation.

Karamanis (2000), Karamanis et al. (2001) performed an investigation on the pulsating flow behaviour of the mixed flow turbine at its inlet and exit by using the Laser Doppler Velocimetry (LDV) visualization method. This investigation reaffirmed that the turbine performance and its flow characteristics under pulsating flow conditions deviate from their steady state counterparts, and that the performance map formed a significant hysteresis around the steady state curves. As the flow frequency increased, the hysteresis loop was found to be smaller in size - an indication that flow conditions within the turbine tend to steady flow values. Karamanis (2000) also observed that the total-to-static instantaneous efficiency loop obtained at the stator inlet was wider than that at the rotor inlet. It was found that the intermediate inlet and exit velocity triangles showed significant variation with time. By comparing the velocity triangles under both steady and unsteady conditions, Karamanis et al. (2001) concluded that the quasi-steady assumption became invalid at the beginning and end of the pulse cycle.

In assisting with the characterization of the difference between steady and unsteady turbine performance, Szymko (2006) developed a uniquely designed permanent magnet eddy current dynamometer capable of extracting up to 60 kW of turbine power output. This dynamometer was heavily cooled during operation and was designed to react on the torque produced by the turbine, therefore enabling its pure aerodynamic efficiency to be determined without interference from mechanical losses. A further benefit of this new dynamometer was the ability to provide
wide turbine performance maps almost 300% wider than a standard turbocharger map obtained using a compressor as the loading device. This eliminated the need to perform map extrapolation in order to compare the steady state turbine performance with the unsteady performance. Szymko (2006) attempted to characterize the operating regime of the turbine under pulsating flow conditions by the introduction of the normalised Strouhal number. Szymko (2006) indicated that the turbine would behave in a quasi-steady manner if the normalized Strouhal number, St.* < 0.1, in a ‘filling and emptying’ manner for St.* > 0.1 > St.(p)*, (St.(p)* being the pressure modified Strouhal number) and in a ‘wave action’ manner for St.(p)* > 0.1. Meanwhile, this study also implied that at a pulse frequency of 20 Hz, the cycle-averaged efficiency was generally 6% lower than its quasi-steady counterpart but that this discrepancy reduced with increasing pulse frequency. Furthermore, the experimental study of Szymko (2006) also showed that the isentropic power under pulsating flow was larger than steady flow conditions.

Hakeem et al. (2007) performed an investigation of steady and unsteady turbine performance by comparing the effect of volute geometry of different designs as shown in Figure 1.22, indicating that the use of a volute with larger swallowing capacity resulted in higher isentropic efficiency at high velocity ratios. This study also found that the design of the new volute had reduced the turbine’s sensitivity to rotational speed and was expected to improve the performance under pulsating flow operations. Again, for the unsteady investigations, Hakeem et al. (2007) noted that the turbine instantaneous performance and flow characteristics deviated substantially from their steady state values and that the quasi-steady assumption that was normally used in the performance assessment of turbine operation under pulsating flow conditions was inadequate. The cycle-averaged efficiency was also found to be higher than the steady state value under all tested operating conditions.

Rajoo and Martinez-Botas (2008b) conducted the experimental evaluation of a variable geometry mixed flow turbine under steady and pulsating flow using both straight and lean nozzle vanes and at different vane angle settings. The lean nozzle was designed to have 40° lean stacking from the axial direction, in order to match the 3D configuration of the mixed flow turbine leading edge. Meanwhile, the straight
vane was constructed for comparison purposes in order to investigate any benefits of the new lean vane design. The lean vanes indicated a clear improvement in efficiency at velocity ratios higher than 0.65, using a high vane angle setting (almost closed position). In the pulsating flow experiment, Rajoo and Martinez-Botas (2008b) revealed that the range of mass flow parameter and pressure ratio were higher for straight vanes than for lean vanes. Furthermore, the cycle-averaged efficiency in the straight vane configuration is found to be higher than the lean vane design. Moreover, the comparison to the equivalent quasi-steady was better for the straight vane configuration compared to the lean vane.

Copeland (2009), Copeland and Martinez-Botas (2008) and Copeland et al. (2009, 2010, 2011) in their research investigated the performance of a double-entry turbine under pulsating flow as well as its deviation from the quasi-steady assumption. Since a double-entry turbine was used, steady-state equal and unequal admission experiments were also conducted. They later demonstrated that the unsteady efficiency of the turbine suffered most at the lowest frequency. Furthermore, the quasi-steady assumption displayed the ability to follow a similar performance trend to that under pulsating flow condition but could not replicate the magnitude of the unsteady efficiency drop with increasing frequency. In other words, the quasi-steady assumption consistently under predicted the magnitude of mass flow, especially at lower frequencies. Therefore, it can be concluded that the quasi-steady assumption...
only reflects the rough trend of the unsteady performance and should not be used as a main tool to obtain quantitative predictions of turbine efficiency with unsteady admission. In addition, they also suggested that lower flow frequencies resulted in greater time available for the turbine wheel to swallow the mass introduced over a pulse cycle, therefore encouraging a better flow field in the volute, while the opposite occurs for higher flow frequencies.

Another major issue in the analysis of turbine performance under pulsating flow conditions is the phase-shifting method. Since the pulse travels around the turbocharger turbine in a finite time, an appropriate phase-shifting technique has to be applied in order to ensure that the ratio between the isentropic power available at the turbine inlet and the actual power extracted at the turbine shaft is calculated for the same phase instant. Over the years, different phase-shifting techniques were utilized by different researchers. For instance, Dale and Watson (1986) and Arcoumanis et al. (1995) used the sonic velocity for phase-shifting meanwhile Winterbone et al. (1991) and Baines et al. (1994) used the bulk flow velocity on which to base their phase shifting adjustment. Recent studies by Rajoo and Martinez-Botas (2007) and Szymko et al. (2005) indicated that the sum of sonic and bulk flow velocities can also be used as the pulse travelling speed. Further discussion on different phase shifting methodologies as well as their impact on instantaneous efficiency calculation is discussed in Chapter 5.

1.2.5 Numerical work on pulsating flow turbine characteristics

In the 1990s, Chen et al. (1996) conducted a one-dimensional simulation of a mixed flow turbine under pulsating inlet conditions by considering the volute to be a tapered duct of a certain length, and solved the 1-D unsteady gas dynamic equations. The rotor was simulated by a quasi-steady flow method where the rotor was treated as if it behaves exactly as under steady flow conditions at every instant in time. Furthermore, the correct volume of the volute casing was specified by properly defining the flow area of the pipe. Numerous loss terms, namely circumferential variation loss,
casing friction loss, incidence loss and other losses were modelled, improving upon the previous work by Chen and Winterbone (1990). For the inlet boundary conditions, the total pressure was required. Chen et al. (1996) found that the application of the quasi-steady flow method always underestimated the turbine swallowing capacity. Despite that, the model developed by Chen et al. (1996) indicated that the prediction of instantaneous turbine power was in good agreement with experimental data.

The assumption of quasi-steady flow in the rotor was continued by a one-dimensional unsteady flow simulation developed by Abidat (1991), Abidat et al. (1998). The unsteady one-dimensional flow equations in this work were solved in the volute by a finite difference method using a four-step explicit Runge-Kutta scheme and the volute exit conditions were provided by the previous quasi-steady assumption in the rotor. For the inlet boundary conditions, only the total inlet pressure was varied with time whereas other variables such as the total inlet temperature and exit static pressure area were assumed to be constant during the cycle. Abidat et al. (1998) also investigated the effect of the frequency and the amplitude of pulse performance of the mixed flow turbine in their work by assessing the departure of the unsteady characteristic from the equivalent quasi-steady condition. The results obtained from this work indicated an improved agreement as compared to Chen et al. (1996). Furthermore, the predictions of averaged mass flow rate and average turbine output power for different frequencies and amplitudes showed good agreement between the unsteady flow model and the one obtained through a quasi-steady assumption. Therefore, Abidat et al. (1998) concluded that the quasi-steady flow analysis was sufficient for a good approximation of the two parameters (averaged mass flow rate and averaged turbine output power).

Costall et al. (2006) and Costall (2007) developed a one-dimensional model to simulate pulse flow in a single and twin entry turbocharger turbine with additional enhancements over the work of Chen et al. (1996) and Abidat et al. (1998). The configurations of the computational domain of both types of volute are shown in Figure 1.23. In applying the unsteady boundary condition at the domain inlet, Costall (2007) utilized a “transmissive” boundary condition that replicates the ex-
Experimental pulse while acting as a non-reflecting boundary condition. This prevents waves emanating from the interior reflecting at this plane, since in reality the experimental domain extended further upstream. As one would expect, the exit boundary condition is simply an open end boundary condition that applies the stagnation pressure and temperature of the surroundings. Costall (2007) utilized the two-step Lax-Wendroff scheme with the total variation diminishing flux limiter resulting in a second-order, conservative, shock-capturing finite difference solver.

The single entry model was validated against experimental results at 60 Hz flow frequency as shown in Figure 1.24. The simple single entry model was able to successfully predict mass flow and power (see Figure 1.24). As for the twin entry...
model under full admission conditions, there was no clear winner between the single and twin entry models taking both mass flow and power predictions into account, hence the single entry would be favoured given its simplicity. Costall et al. (2009) suspected that incorrect selection of the loss coefficient is the primary source of error that resulted in under prediction of both parameters. Another important outcome from the work of Costall et al. (2009) was that there are a few potential improvements that could further improve the prediction capability of a 1-D model. For instance, the effect of moving the single rotor inlet station relative to the tongue could be further investigated and that adding more junctions to permit further locations for rotor inlet flow and limb interactions could also be studied.

Romagnoli and Martinez-Botas (2007, 2011) took advantage of the wide range experimental turbocharger turbine map obtained at Imperial College London as a validation of their one-dimensional model of nozzled and nozzleless mixed-flow turbines. They performed a meanline calculation to predict turbine performance under steady state conditions and carried out a breakdown of all aerodynamic losses. However, the prediction of turbine performance across the entire extended experimental map was challenging. Romagnoli and Martinez-Botas (2007, 2011) found that the existing loss models failed to provide an accurate prediction in the high velocity ratio region of the map where an incidence factor had to be included and calibrated. Furthermore, the breakdown analysis of losses indicated that the incidence loss accounted for the largest portion of the energy dissipated (except, as might be expected, at the peak efficiency). At this condition, the passage loss was higher due to the high level of kinetic energy; the clearance and disc friction losses only accounted for less than 3% of the overall energy dissipated.

Recent work by Chiong et al. (2011) involved five models with increasing complexity in order to predict the instantaneous performance of a twin-entry turbine under full admission 60 Hz pulsating flow. These models consisted of several components: the inlet duct, the volute, the turbine wheel and the turbine exit pipe as shown in Figure 1.25. The variations of geometrical properties of these components were studied in detail in order to obtain the most accurate predictions. Chiong et al. (2011) concluded that the varying area ducts has the effect of magnifying the sec-
ondary fluctuations in mass flow rate prediction. Furthermore, Chiong et al. (2011) indicated that more experimental data were necessary in order to further improve the model. Despite that, this study contributed to a deeper understanding of the assumptions involved in the development of a 1-D model of a turbine subjected to pulsating flow inlet conditions.

Figure 1.25: Schematics of models with increasing complexity (Chiong et al., 2011)

The application of 1-D models for turbocharger turbines are not only limited to the performance prediction of the turbine. Many have adopted this modelling methodology and employed it as a monitoring or a diagnostic tool for a relatively large turbocharging system. As an example, Barelli et al. (2013) developed a 1-D
model of a CHP plant to simulate failure conditions of the turbocharged air group in order to avoid plant stoppage. This work particularly showed that it was indeed possible to expand the scope of 1-D turbocharger modelling to the larger scale as long as the model was constantly validated in order for it to remain viable.

There have been multiple attempts to model the full stage turbocharger turbine using a 3-D CFD approach either to study, to predict or to design a completely new turbine concept. One of the early attempts was that of Lymberopoulos et al. (1988) which they utilized a simplified quasi-three-dimensional solution of the Euler equation. This effectively meant that the radial and tangential components of velocity were fully solved, but the axial component was only treated to simulate the mixing of the two streams. The numerical model of the single volute was validated with experimental results which indicated good agreement. However, the validation process for the twin entry case was rather limited due to the employment of a coarse grid in the axial direction. Lymberopoulos et al. (1988) discovered that there were significant variations in flow properties around the exit circumference of the volute, particularly at the region close to the tongue due to the recirculation flow.

An example of the full 3-D CFD analysis to assist the development of a turbocharger turbine was demonstrated by the work of Barr and Spence (2008). In this work, CFD has been used extensively in the development of the back swept blading for a radial turbine with aimed to improve the off-design efficiency particularly at low velocity ratio (high pressure ratio) operation. The primary finding where the newly designed turbine indicated 2% more efficiency than its counterpart was clearly supported by extensive flow field analysis that showed improved flow features (less flow separation) at the rotor inlet. Furthermore, the CFD simulation of Barr and Spence (2008) also indicated that the back swept radial turbine performed less efficiently at high velocity ratio. They later justified the design concept since the improvement that a back swept blading brought at low velocity ratio outweighed the performance deficit recorded at high velocity ratio. Moreover, in this work, CFD was used alongside a preliminary 1-D method as well as finite element analysis in order to complete the whole design process. Barr et al. (2009) later extended their work to include experimental validations. However, the experimental data obtained
could only cover half the operating points and the lowest velocity ratio that was predicted to have the most significant improvement could not be captured. Despite that, the advantage of back swept blading was still visible. As the back swept blading inevitably altered the geometrical properties of the radial blading, one would expect that the stress would be too high since this turbine was designed for high pressure ratio conditions. However, the finite element analysis conducted by Barr et al. (2009) indicated that the stress or fatigue risk of the backswept rotor should not be worse than for the existing radial rotor.

![CFD calculations](image)

**Figure 1.26:** CFD calculations by Barr et al. (2009) showing a smaller separation region in the backswept rotor (b); SS = Suction Surface, PS = Pressure Surface

As a turbocharger turbine often operates at off-design conditions, Walkingshaw et al. (2010) conducted a CFD analysis in a turbine that operated at velocity ratios as low as 0.34. They indicated that the tip clearance vortices that emerged from the vanes could also have favourable effect on the circumferential incidence angle distribution. Walkingshaw et al. (2010) indicated that the potential reason for flow distortion was more likely due to the wake flow originating from the vane trailing edge. However, within the blade passage, they observed strong separation on the suction side surface attributed to poor inlet flow alignment and that the flow struggled to reattach to the suction surface and therefore resulted in negative blade loading. However, the simulation works that were carried out have not been verified by experimental results. Walkingshaw et al. (2010) later claimed that a radial rotor that was not restricted to the radial fibre design could improve the flow behaviour.
Chapter 1. Introduction and literature review

during high pressure ratio operation of the turbine.

Tamaki et al. (2008) conducted an investigation about the effect of the tip clearance of a variable area nozzle on the performance of a radial turbine. This work was properly validated by experimental data (see Figure 1.27). It was deduced from this simulation that the tip leakage flow of the nozzle was significant only during the operation where the nozzle is at its smallest opening (see Figure 1.27(a)). As the nozzle area increased, the effect of tip clearance weakened as indicated in Figure 1.27(b). Tamaki et al. (2008) indicated that the two main flow features that were observed downstream of the vane rows were the wake flow and the leakage vortex. During operation where the vanes were set to their smallest opening, the flow angle calculations downstream of the vanes indicated that the wake flow quickly diminished and the leakage vortex dominated. This leakage vortex further distorted the total pressure distribution and flow angle which in turn caused mixing loss downstream of the nozzle vanes and the turbine performance to further deteriorate. On the other hand, as the vanes were set to their largest opening, only the wake flow was detected and the leakage vortex seemed to be restricted to the region very close to the shroud wall. Furthermore, Tamaki et al. (2008) also pointed out that the ratio of the leakage flow rate to the total flow rate was almost proportional to the clearance at the same nozzle opening. This work also addressed the issue that the leakage losses that occurred in the nozzle vane stage potentially have a negative effect on overall engine performance especially during periods of acceleration where the vane opening is at its smallest.

Although there are many CFD works conducted to simulate the turbocharger turbine and further its development, relatively little work has been done to simulate and analyse its pulsating behaviour. Perhaps the earliest successful pulsating flow CFD simulation work was that of Lam et al. (2002). The main aim of this work was to demonstrate the capability of CFD to provide a sufficient description of the flow within the nozzles and the turbine passage. Lam et al. utilized the Multiple Rotating Frames (MRFs) method which is also known as the ‘frozen rotor’ approach, that assumes no relative movement between stationary and rotating frames of reference during the simulations. Lam et al. (2002) also indicated that the MRFs method...
Figure 1.27: Effect of the tip clearance on flow angle distribution at (a) smallest vane opening and (b) largest vane opening (Tamaki et al., 2008)

was only valid for steady state conditions where the flow at the interface between stationary and rotating domain is assumed be relatively uniform. For unsteady calculations, the assumption of relatively uniform flow was only valid if the time scale of the transient effect (pulsating frequency) is relatively large compared to the time scale of the rotating body (rotor rotation). Therefore Lam et al. (2002) justified this method for their unsteady calculation since they applied a pulse frequency of 53.33Hz which was much smaller than the turbine frequency of 2267Hz. This work also highlighted a few difficulties in order to perform and evaluate unsteady turbine performance. One of the issues was the difficulty of getting the simulation to converge properly therefore only qualitative comparisons could be made. Furthermore, the complex geometry of the turbine itself presented a difficulty in defining the exact entry point of unsteady flow at the rotor inlet. Despite that, the work of Lam et al. (2002) proved that it was possible to further understand the complexity of pulsating flow by means of full 3-D CFD.

An improved CFD work by employing a sliding interface between the stationary and rotating domain was carried out by Palfreyman and Martinez-Botas (2002). In addition, this work also employed a single passage with a refined mesh and periodic boundary conditions assuming that the flow would behave identically for every passage. The simulations were extensively validated by the Laser Doppler Velocimetry measurement work of Karamanis (2000), although there was a localized discrep-
Chapter 1. Introduction and literature review

ancy due to the tip leakage vortex. The work of [Palfreyman and Martinez-Botas (2002)] allowed details of the flow field to be captured and analysed. Furthermore, this work also indicated that the main source of secondary flow was the tip clearance flow. In addition, [Palfreyman and Martinez-Botas (2002)] took advantage of their CFD technique to perform comparisons between the flow fields within mixed flow and radial turbines. It was found that the mixed flow turbine has an advantage over the radial turbine in that the mixed flow turbine has a smaller region of entropy generation close to the suction surface tip. This was attributed to a large range of vorticity concentrated near the shroud wall of the radial turbine due to the sudden turning of the streamlines in the meridional plane from the radial to axial direction.

[Chen et al. (2008)] conducted another CFD simulation in order to investigate the flow structure within the rotor stage of a highly loaded turbine (high pressure ratio operation). This investigation utilized a single passage simulation, allowing greater mesh density (close to 800k nodes) thus more flow details could be made visible for analysis. [Chen et al. (2008)] also found that the tip leakage only contributed a small proportion of the vortex that existed within the blade passage. They indicated that only a little fluid was driven from the pressure side to the suction side through the tip gap due to the high relative velocity of the shroud. This finding seems to be contradictory to that of [Palfreyman and Martinez-Botas (2002)] who observed a significant contribution of the tip leakage to the overall generation of the secondary flow. However, the major difference in these works was that [Chen et al. (2008)] focused on the simulation of the turbine operating under high pressure conditions whereas [Palfreyman and Martinez-Botas (2002)] were simulating the turbine at its design condition. [Chen et al. (2008)] also concluded that the secondary flow structure within the turbine passage is highly three-dimensional as an inevitable result of the tip leakage vortex, the horseshoe vortex, and the corner vortex that dominated the inducer region.

[Palfreyman and Martinez-Botas (2005)] later extended their work by including the pulsating pressure profile as the boundary condition at the domain inlet where all the components are modelled (see Figure 1.28). The main feature of this simulation is that the interface between stationary and rotating domain was defined.

1.2. Energy recovery through turbocharging
Chapter 1. Introduction and literature review

1.2. Energy recovery through turbocharging

Figure 1.28: Inlet, exit and sliding boundary conditions, (Palfreyman and Martinez-Botas, 2005)

Figure 1.29: Pressure at the center of the volute at given azimuth angles (a) experiment, (Karamanis, 2000) and (b) predicted (Palfreyman and Martinez-Botas, 2005)
as a ‘sliding-plane’ where there were relative motion between the domains on both sides of the interface at each time steps. Moreover, [Palfreyman and Martinez-Botas (2005)](#) indicated that the employment of this method ultimately improved the model capability to capture the hysteresis loop of mass flow rate and efficiency appropriately without the degree of damping observed by [Lam et al. (2002)](#). The predicted static pressure at several locations within the domain also compared well with the experimental data (see Figure 1.29). The results of this work indicated that the flow field in the turbine passages is highly disturbed, and that the effect of the blades passing the volute tongue had most effect in the inducer region. Furthermore, [Palfreyman and Martinez-Botas (2005)](#) also concluded that the poor flow guidance observed particularly at the inlet and exit of the turbine was due to the assumption of quasi-steady conditions during the design stage of the particular turbine rotor.

A study of the effect of variation in flow profile at the inlet boundary of the computational domain on the turbine performance was conducted by [Hellstrom and Fuchs (2008)](#). This simulation work revealed that the existence of highly disturbed flow at the turbine inlet could have the effect of reducing turbine shaft power output. However, the work of [Hellstrom and Fuchs (2008)](#) was not properly validated due to difficulties in obtaining the actual inlet disturbance during experimental measurements. Nevertheless, this investigation yielded important qualitative information regarding the mechanisms for the reduction of shaft power. Firstly, the perturbations at the inlet resulted in the reduction of the turbine power output since the flow velocity is reduced before entering the turbine wheel due to excessive pressure losses. Secondly, the induced swirling increased oscillations in the secondary flow at the inlet of the rotor due to the existence of counter-rotating vortices in the volute. This behaviour ultimately resulted in unfavourable radial velocity distributions, in turn reducing the power generation capability of the turbine. Furthermore, their investigation regarding the tip clearance revealed that the strength of the tip vortices only has a secondary effect on the shaft power. This observation matched the observation made earlier by [Chen et al. (2008)](#).

[Hellstrom and Fuchs (2009)](#) later extended their work to include a relatively large domain (see Figure 1.30) of an exhaust manifold upstream of the turbine housing in
order to investigate engine-realistic flow perturbations at the volute inlet. Despite lacking the experimental data for validations, Hellstrom and Fuchs (2009) pointed out that the shaft power is lower during the acceleration phase of the mass flow compared to the deceleration phase. They also claimed that the geometry upstream of the turbine has a large effect on turbine behaviour and that the minimal phase shift of only $4^\circ$ was recorded possibly due to the large manifold volume acting as a reservoir. Furthermore, a particularly interesting observation was that the instantaneous mass flow rate experienced a substantial drop right where it was expected to peak. This behaviour was associated with the back flow that occurred due to the interaction of multiple exhaust manifold branches. Moreover, Hellstrom and Fuchs (2009) indicated that the axial velocity leaving the exhaust manifold deviated substantially from a uniform pipe flow and that strong secondary flows were present. However, these observations were limited only to the particular configuration of the exhaust manifold in question and therefore the data obtained would not be valid for comparison against other turbines.

More recent work on pulsating flow 3D simulation was conducted by Copeland et al. (2012) on the double entry turbine. This work attempted to characterize the level of ‘unsteadiness’ within the turbocharger turbine stage by defining a parameter, $\Lambda$, that represents the ratio of the time-averaged rate of change of the mass flow

Figure 1.30: Numerical domain used by Hellstrom and Fuchs, 2009
within the domain to the time-averaged through flow mass. Therefore the flow inside
the domain is considered steady when the Λ value is 0 and that it is unsteady
when Λ is 1. This work also hinted that the rotor stage is not wholly quasi-steady
but is insignificant enough as compared to the volute stage. The definition of Λ is
useful since it depends not only on the flow frequency but also its amplitude. As
such, this parameter is employed in this thesis to aid analysis in Chapters 4 and 5.

Building on the work of Copeland (2009), Newton (2014) investigated different
vane geometries to obtain the best arrangement to suit unsteady flow operation,
through a combination of numerical and experimental work - the 3D models were
fully validated with excellent agreement in instantaneous mass flow predictions. Per-
haps the most interesting part of this work is the comparison between steady and
unsteady loss generation. Newton (2014) concluded that the overall entropy gener-
ation in the pulsating case was 1.66% higher than the corresponding steady state
condition, and could be attributed primarily to an increase in entropy generation
in the nozzle-rotor interspace region. Newton (2014) also supported the findings of
Copeland et al. (2012) that the rotor is not completely quasi-steady but remains
very close to it.

Although much CFD work have been carried out over the years, the number
of simulations conducted to investigate turbine performance under pulsating flow
conditions are still very limited. This is likely due to the expensive computational
cost as well as the difficulties in obtaining experimental data for validation purposes.
Without further full 3D simulations to investigate the phenomena, it is unlikely
that the design of turbocharger turbines that take advantage of pulsating flow will
improve. With the promising advantage of the mixed flow turbine to operate more
efficiently at high pressure ratio, and with the addition of the novel guide vanes that
geometrically matched the turbine inlet (Rajoo, 2007), it is necessary to investigate
in greater detail the flow field within this particular turbine stage. The availability
of experimental data as well as the advancement of computing facilities (e.g. parallel
processing across multiple cores) have prompted the commencement of the current
research project. To the author’s knowledge, such simulations of a single entry,
leaned vane, mixed flow turbine do not yet exist and as such suggest the novelty of

1.2. Energy recovery through turbocharging
the current research.

1.3 Research objectives

The main objectives of this research are:

- To construct a validated numerical model that is able to simulate turbine operation under pulsating flow conditions
- To perform detailed analysis on the flow behaviour within the turbocharger turbine stage under pulsating flow conditions
- To develop and test a highly efficient turbine stator concept in both steady and pulsating flow environments

1.4 Research scope

To achieve these objectives, this research focuses on the simulation of a single entry, lean vaned, mixed flow turbine. The selected turbine, designated ‘Rotor A’, was originally designed by Abidat (1991) at Imperial College London. This turbine is used together with the previously modified Holset H3B volute to accommodate the lean vanes designed by Rajoo (2007). The simulation work also employs rigorous validation procedures in order to gain confidence in the accuracy of the model. As there have been arguments with regards to the most appropriate phase shifting procedure, this work also assesses multiple phase shifting techniques and explains their advantages and disadvantages. Furthermore, this research provides a detail analysis of the flow field behaviour within the turbocharger turbine stage for both steady and unsteady simulations. It also brings together direct comparisons of the flow field between steady and unsteady inlet flow conditions at a similar pressure level. Moreover, this work presents comparisons of flow angle distributions at the turbine leading edge between different flow frequencies.
The next stage of this research concentrates on the development of a novel concept which involves aggressive reduction of the number of vanes. This requires experimental work to be done in order to prove the effectiveness of the new design in both steady and unsteady conditions, using both vaneless and vaned (original geometry) geometries as baselines for comparison.

1.5 Thesis outline

This thesis consists of six chapters:

CHAPTER 1 Introduction and Literature Review This chapter presents a thorough research background as well as its aims and objectives. A clear motivation for the current research that highlights the worldwide $CO_2$ emissions as a direct contribution from the automotive sector that could be as high as 22%, is discussed. A comprehensive review of the work by previous researchers that are pertinent to this research are included to establish a clear research particularly in the area of turbine wheel performance characteristics under pulsating flow conditions.

CHAPTER 2 Experimental Methodology The experimental setup for both steady and pulsating flow conditions are described in detail within this chapter. Since the experimental rig available for turbocharger testing at Imperial College London is a cold flow facility, the principal of similarity is also explained. All measurements of parameters that are required to obtain actual and isentropic power as well as a few other additional parameters for model validation are described in detail, including an uncertainty analysis.

CHAPTER 3 Simulation Methodology This chapter aims to provide a background on the Computational Fluid Dynamics (CFD) approach taken during the current research. A brief description of Reynolds-Averaged Navier-Stokes (RANS) methods as well as the associated closure problem is included. Furthermore, the employment of the k-epsilon turbulence model to enable closure of the RANS equations, together with its advantages and limitations are detailed. Subsequently, this chapter provides
a thorough description of the development of a full stage three-dimensional CFD model from the geometry creation through the generation of computational results.

CHAPTER 4 Steady flow turbine operations This chapter begins with the validation exercises using the turbine efficiency and its swallowing capacity. The subsequent analysis details the flow field behaviour within the turbocharger turbine under steady flow conditions. A baseline analysis is conducted at optimum condition and the deviation of flow field when turbine is operating at off-design condition is presented.

CHAPTER 5 Pulsating flow turbine operations Following chapter 4, this chapter details the comparison between steady and pulsating turbine operations. This chapter also presents the flow field analysis at different flow frequencies with the particular focus on the flow angle distributions across the stator and rotor inlet circumference. The measure of turbine 'unsteadiness' using Lambda parameter, Λ, is presented for all the calculated cases.

CHAPTER 6 Efficiency improvement through aggressive nozzle vanes reduction This chapter presents the background and development of a novel concept of nozzle arrangement, which is experimentally proven to provide similar or better performance than the baseline arrangements (vaneless and full vaned). The efficiency improvements are not only recorded during steady state conditions, but also during pulsating flow.

CHAPTER 7 Closure The final chapter of this thesis provides the overall conclusions of the work as well as further recommendations for future work.

1.6 Research tasks

Task 1 Development of a three-dimensional computational model that incorporates the entire turbocharger turbine stage that matches the geometry of the experimental rig available at Imperial College London. The exact geometrical match between the
numerical model and the actual rig will enable a direct comparison of turbine wheel performance especially for model validation purposes. Particular attention is given to geometrical details such as the tip clearance gap of the rotor and vane blades where the position of the physical device after assembly is modelled rather than its ideal geometry.

**Task 2** Execution of experimental work in order to obtain necessary measurements for model validation. More specifically it ensures continuation as well as validity of the previously available data for a similar turbine configuration. The experimental uncertainties are properly recorded during execution of this task therefore any inconsistencies during the validation procedure with the numerical predictions can be traced accordingly to either the model or an issue with the experimental data itself.

**Task 3** Analysis of the available experimental and numerical data for both steady and pulsating flow turbine operation. This task involves a large database mainly of computational results for pulsating flow conditions that requires almost 4TB of storage in total. Eight cases of pulsating conditions - two speed lines (30000 rpm and 48000 rpm) and four frequencies (20, 40, 60 and 80Hz), were calculated. It is therefore a lengthy process and the results obtained from this task contribute to a significant proportion of this thesis.

**Task 4** Development and experimental evaluation of a novel stator concept involving aggressive reduction of the number of nozzle vanes. The theoretical development of the concept emerged from detailed analysis of the massive amount of CFD information from Task 3. This task requires modifications to be made only to the vane blade number and their arrangement. Meanwhile, other components such as the inlet duct, the volute as well as the rotor remain the same in order to be able to directly compare the performance of this new concept to the original design.
Chapter 2

Experimental methodology

2.1 Introduction

The turbocharger facility available at Imperial College London was originally developed by Dale and Watson (1986). Since then, the facility has undergone many modifications by Baines and Lavy (1990), Abidat (1991), Arcoumanis et al. (1995), Hakeem et al. (2007), Karamanis et al. (2001) and Pesiridis (2007) with the primary aim to provide accurate measurement of highly pulsating flow parameters, as well as flow visualization and to accommodate different type of turbine wheel such as mixed flow turbine. Perhaps, the most important advancement of the facility is the addition of the Eddy Current Dynamometer developed by Szymko (2006) to measure the turbine power output. This enables up to three times wider turbine operating range map to be produced as compared to the use compressor as the loading device.

This chapter intends provides a detailed description of the turbocharger test facility arrangement and instrumentation that are pertinent to this research project. Furthermore, certain fundamental aspects with regards to similarity theory and dimensional analysis are also discussed. Subsequently, in order to gain enough confidence about the measured parameters, uncertainty analysis is also discussed in this chapter.
2.2 Dimensional analysis and similarity approach

In the pursuit of understanding turbine performance parameter, it is useful to be able to compare turbine performances regardless of its dimensions and flow conditions. Therefore, all the physical parameters associated with turbine operation need to be reduced to either completely non-dimensional or semi-dimensional. With reference to Watson and Janota (1982), the relevant parameters that are directly involved in the definition of turbine efficiency and mass flow rate are:

\[
\begin{align*}
\text{efficiency, } & \eta \\
\text{mass flow rate, } & \dot{m}
\end{align*}
\]

\[
\begin{align*}
&= f \\
&\left\{ \begin{array}{l}
\text{static outlet pressure, } P_5 \\
\text{total inlet temperature, } T_{01} \\
\text{turbine rotational speed, } N \\
\text{turbine inlet diameter, } D \\
\text{gas constant, } R \\
\text{specific heat ratio, } \gamma \\
\text{dynamic viscosity, } \mu
\end{array} \right\} 
\end{align*}
\] (2.1)

The parameters above can be reduced further to a set of non-dimensional groups using the principles of the Buckingham $\pi$ Theorem. The resulting function is,

\[
\frac{\dot{m}\sqrt{RT_{01}}}{P_{01}D^2}, \eta = f \left( \frac{ND}{\sqrt{RT_{01}}}, \frac{P_{01}}{P_5}, \frac{\dot{m}}{\mu D}, \gamma \right) 
\] (2.2)

The Reynolds number of the gas does not contribute to significant change in machine performance, and since the rotor inlet diameter for the current test rig is constant, both parameters can be excluded from the function. Furthermore, the specific heat ratio, $\gamma$ and gas constant, $R$ are always predefined according to the type of working fluid. These result in further reduction of original function to,

\[
\frac{\dot{m}\sqrt{T_{01}}}{P_{01}}, \eta_{ts} = f \left( \frac{N}{\sqrt{T_{01}}}, \frac{P_{01}}{P_5} \right) 
\] (2.3)

where $\dot{m}\sqrt{T_{01}}$ and $\eta_{ts}$ are known as quasi-nondimensional mass flow parameter and
total-to-static efficiency respectively. In addition, it is common practice to plot the efficiency against velocity ratio \cite{Dale86}. Velocity ratio is defined as the ratio between the blade tip speed, $U$ and the isentropic velocity, $C_{is}$,

$$Velocity\ Ratio, \ VR = \frac{U}{C_{is}} \quad (2.4)$$

The isentropic velocity, $C_{is}$ is defined as the velocity that the working fluid would achieve if the isentropic expansion over a particular pressure ratio occurs across the turbine stage. Therefore, for the current research, the turbine performance maps are evaluated using the mass flow velocity vs. pressure ratio plot and the efficiency vs. velocity ratio plot.

In order to conduct the test in cold-flow condition, few parameters are required to be scaled to reproduce the actual performance during hot engine operation. The similarity approach indicated by Gaussman in 1972 enables direct comparison of hot and cold flow testing condition by matching the mass flow parameter and speed parameter accordingly. For similar pressure ratio between actual and testing condition, the similarity approach provides the corresponding mass flow rate and turbine speed for cold-flow testing

$$\left( \frac{\dot{m}\sqrt{T_{01}}}{P_{01}} \right)_{test\_rig} = \left( \frac{\dot{m}\sqrt{T_{01}}}{P_{01}} \right)_{actual} \quad (2.5)$$

$$\left( \frac{N}{\sqrt{T_{01}}} \right)_{test\_rig} = \left( \frac{N}{\sqrt{T_{01}}} \right)_{actual} \quad (2.6)$$

By following the above discussion, the necessary parameters required for calculation of the turbine performance are the mass flow rate, inlet total pressure, inlet total temperature, turbine speed and torque. For the steady flow experiment, the ambient pressure is assigned as the static pressure at turbine outlet.
2.3 Test-Rig arrangements

The turbocharger test facility at Imperial College London assembly consists of a series of components, starting with the supply air compressor, main and safety valves, heaters, orifice plate, pulse generator, measurement plane, turbine assembly and eddy current dynamometer. The 3-D representation of the test rig is shown in Figure 2.1. The air supply for the test rig originated from three Ingersoll Rand screw-type compressors, with maximum capability to deliver air up to 1 kg/s at maximum pressure of 5 bar (absolute). The air is then filtered through a three-stage cyclone. Downstream of the filters are two computer-controlled valves where one is used for regulating the mass flow into the turbine and the other acts as a safety valve. The shutdown of the safety valve is triggered by activation of ‘guillotine valve’ in the event of emergency. Subsequently, the airflow is directed through a set of heaters where the temperature is raised and maintained during testing in order to prevent condensation of the air during expansion in the turbine.

Immediately downstream of the heater, the warm airflow is branched into two 81.4mm diameter pipes, namely the outer limb and the inner limb based on their relative position as illustrated in Figure 2.1. These pipes stay separated up to the measurement plane, thus allowing not only single entry turbine to be tested, but also twin and double entry turbines. Orifice plates were installed on both limbs in order to measure the airflow mass flow rate. The airflow is then directed through the pulse generator originally designed by Dale and Watson in 1986. The pulse generator consists of two specifically designed rotating cut off plate, namely the chopper plate, in order to replicate the actual shape of pulsating pressure profile coming out from the exhaust valve of an engine. These rotating plates are driven by a computer-controlled motor that allows users to control the airflow pulsating frequency that goes into the turbine whether the pressure profile on both limbs are in-phase of out-of-phase. Downstream the pulse generator is the measurement plane where all averaged and instantaneous flow parameters are recorded. These include high response pressure transducers, static pressure tappings, thermocouples and hotwire traversing system for instantaneous mass flow measurement. The con-
necting duct then connects the measurement plane and the volute inlet. This duct could be a converging duct to converge the flow from two separate limbs into a single entry volute as used in this research, or it could also keep the flow separated into the multiple entry volute. The turbine assembly is attached to an eddy current dynamometer capable of extracting power up to 60 kW. The dynamometer is heavily water-cooled in order to quickly dissipate the heat generated due to power absorption from the turbine. Apart from its primary purpose to measure the turbine torque, the dynamometer is also equipped with the optical speed sensor for turbine speed measurement.

![Figure 2.1: Schematic of turbocharger test facility at Imperial College London](image)

### 2.3.1 Test-Rig instrumentation

As previously explained on section 2.2, the required parameters for calculation of the turbine performance are:

- Mass flow rate
- Pressure
• Temperature

• Rotational speed, and

• Power

All of these parameters are measured during both steady and unsteady testing. This section intends to provide detailed explanation about measurement strategy that allows every one of these parameters to be determined.

2.3.1.1 Mass flow rate

*Steady flow condition.* The mass flow rate for both limbs during steady state testing are measured using a McCrometer v-cone flow meter (see Figure 2.2). This is a differential type flow meter. Therefore, it requires measurement of the differential pressure between the high and low pressure ports and the measurement of the absolute pressure in the high pressure port. The measurement of gas temperature is also required. For this purpose, the differential pressure is measured using a Siemens Sitrans P DSIII differential pressure transmitter. This transmitter sent the electronic signal that varied linearly with pressure. This signal was fed directly to the analogue input Fieldpoint module, FP-AI-110. Meanwhile, the absolute pressure was measured using the scanivalve and the air temperature was measured using an E-type thermocouple. Equation 2.7 is used to calculate the final mass flow rate.

![Diagram](image.png)

Figure 2.2: V-cone flow meter geometry, installation, operation and maintenance manual, McCrometer, 2013
\[ \dot{m} = \sqrt{\rho F_a C_d Y k_1 \Delta P} \]  

(2.7)

where \( \rho \) is the density, \( \Delta P \) is the differential pressure across the ports, \( F_a \) is the material thermal expansion factor, \( k_1 \) is the flow coefficient, \( Y \) is the gas expansion factor and \( C_d \) is the discharge coefficient.

The material thermal expansion factor was dependant on temperature and the thermal expansion coefficient of the v-cone material but typically deviated by less than 0.1% from unity. The gas expansion factor, \( Y \), is found by Equation 2.8

\[ Y = 1 - \left( 0.649 - 0.696\beta^4 \right) \frac{U_1 \Delta P}{\gamma P} \]  

(2.8)

where \( U_1 \) is a unit conversion constant prescribed by McCrometer. \( \beta \) is the beta-ratio and is a function of the v-cone geometry

\[ \beta = \sqrt{1 - \frac{d^2}{D^2}} \]  

(2.9)

where \( d \) is the inside diameter of the pipe and \( D \) is the outside diameter of the v-cone. The beta-ratio for the v-cone units used to measure the mass flow rate in each limb was 0.447. The flow constant, \( k_1 \) in Equation 2.7 is a function of the pipe size and the beta-ratio. Both v-cones used during testing had a flow constant of 0.0139. For the discharge coefficient, \( C_d \), the calibration sheets supplied with the v-cones (see Appendix A) indicated that a constant value is to be used for the current testing conditions. Therefore the values of 0.8385 and 0.8472 were used for the two v-cones.

**Pulsating flow condition.** The instantaneous mass flow rate during pulsating flow testing is measured using a constant-temperature hot-wire anemometer (CTA) which is installed at the measurement plane. The CTA used the platinum plated tungsten wire with 10\( \mu \)m diameter coupled to the StreamLine CTA system by Dantec Dynamic. A CTA is effectively a Wheatstone bridge. The working concept of CTA
is such that the appropriate current is applied to balance the voltage of the bridge so that the sensor resistance, thus the temperature is kept constant. The current is controlled very precisely by a servo amplifier. Any high frequency fluctuation that occurs within the airflow that affects the sensor can therefore be detected by combination of both low thermal inertia of the sensor and the high gain of the servo loop amplifier. Therefore, the voltage required to balance the bridge is directly related to the velocity of the flow. King’s Law [King (1914), Lomas (1986) and Brunn (1995)] provided relationship between the voltage and velocity in the form of Nusselt number (heat transfer and wire geometry dependant) and Reynolds number (flow velocity and wire geometry dependant), and the relationship is given in Equation 2.10.

\[ Nu = a + bRe^{0.5} \]  

where a and b are constants. In order to accurately measure the instantaneous mass flow rate, the hotwire is traversed in the plane perpendicular to the flow direction. This covers 36 measurement points where the traversing mechanism and its control system was designed and built in-house in accordance to British Standards (BS1042, 1983).

**Calibration of CTA.** The CTA was subjected to two-stage calibration. For the first stage, the CTA was calibrated using the calibrator provided by Streamline capable of providing jet stream air velocity from 0.02 m/s to 300 m/s. There are two purposes of using this calibrator, one is of course to obtain the constants for King’s Law and the other is to ensure that the tungsten wire is properly attached to the CTA. The latter was achieved by subjecting the hotwire to maximum air velocity for 3 to 5 minutes. The calibration was done by recording the voltage for 15 points at air velocity between 3 m/s to 300 m/s at room temperature of 24°C. The static temperature of hotwire sensor varies with Mach number. To compensate for the difference in the original temperature and the temperature on the testing day, the flow unit was corrected and used in the second stage as the reference temperature.
Chapter 2. Experimental methodology

The transfer function (see Equation 2.11) that was established by King’s Law is used to fit the calibration curve where all the constants can be determined. The typical value for the calibration constants is 4.61, 2.06 and 0.46 for $A_{\text{calib}}, B_{\text{calib}}$ and $n$ respectively.

$$E^2 = A_{\text{calib}} + B_{\text{calib}}(\rho U)^n$$  \hspace{1cm} (2.11)

where $E$ is the CTA voltage (volts), $A, B, n$ are the power law coefficients and $\rho U$ is the mass flux per unit area($kg/m^2 \cdot s$)

The hotwire is then installed into its traversing system and is traversed to its 36 designated positions while the air is flowing. The 36 points readings are then converted into mass flux using Equation 2.11 above. Subsequently all 36 points readings are integrated in accordance to British standard BS1042, 1983. The integrated mass flux was converted into instantaneous mass flow rate by multiplying it with the duct area at measurement plane (see Equation 2.12)

$$\dot{m} = (\rho U)_i \cdot A_{\text{meas, plane}}$$  \hspace{1cm} (2.12)

During unsteady testing conditions, the temperature is known to fluctuate heavily and away from the calibration temperature, thus demanding second stage of hotwire calibration at actual warmed air operation. The StreamLine CTA system was setup to run on constant over-heat ratio during both calibration process and the actual testing (different temperature on both). Therefore the measured raw data were corrected for temperature. The fluid properties which affect the hotwire reading are the Prandtl number ($Pr$), thermal conductivity ($k$), dynamic viscosity ($\mu$), density ($\rho$) and the Mach number ($M$). The effect of these properties with temperature was assessed and used to amend the calibration factors ($A_{\text{calib}}$ and $B_{\text{calib}}$) during unsteady testing. These corrected factors were then integrated into Equation 2.11 and 2.12 to solve for the mass flow rate.

2.3. Test-Rig arrangements
2.3.1.2 Pressure

**Steady flow condition.** The static pressure was measured through static-hole tapping perpendicular to the flow direction at desired points such as orifice plate and measurement plane. These holes are pneumatically connected to a rotary-switch, Scanivalve which was coupled with two strain gauge pressure transducers. One of the transducer (Druck PDCR 23D type) is for high pressure range of $\pm 3.5$ bar (gauge) with uncertainty of $\pm 0.02\%$ FS ($\pm 90$ Pa) while the other transducer (Druck PDCR 22 type) is for low pressure range of $\pm 0.35$ bar (gauge) with uncertainty of $\pm 0.008\%$ FS ($\pm 36$ Pa). A signal amplifier model Flyde FE-351-UA Uni-Amp is used altogether with a signal conditioning model, Flyde FE-492-BBS Mini-Bal to obtain the signal from the transducers. The output of the transducer-conditioner-amplifier modules are connected to the National Instruments FieldPoint analogue input module NI-FP-AI-110 on the data acquisition system. Meanwhile, the rotary switch on the scanivalve itself is controlled via National Instruments FieldPoint digital output module NI FP-DO-401 and its channels are monitored via digital input module NI FP-DI-330. These analogue and digital channels were remotely connected to the control desktop located just outside the test rig cell via Ethernet connection. An in-house built LabView program sequentially switches the Scanivalve channels during the test-logging period to obtain all the relevant static pressure of the flow.

**Pulsating flow condition.** The instantaneous static pressures are recorded at the measurement plane and at the rotor exit. At the measurement plane, two high-response
Schaevitz type P704-0001 strain gauge pressure transducers are used to obtain the static pressure readings for both limbs. The transducers are capable of measuring the pressure range of 0-3.45 bars (gauge) with 0.059% FS maximum deviation. To reduce the pulsation effects (Winterbone et al. 1991), the transducers are mounted close to the duct wall with appropriate tapping geometry. The air passage length from the flush-inner face of the duct to the surface of the sensor is 33mm. The corresponding Helmholtz resonant frequency for the passage length is approximately 2800Hz. Therefore, given that even the maximum pulsating frequency that is experienced during testing is less than 20% of the resonant frequency, the Helmholtz effect on the pressure reading can be assumed to be negligible. At the rotor exit, a SensorTechnics high response strain gauge transducer mode 19C 50P G 7 K is used to obtain the static pressure measurement. It has a range of 0-3.45 bar (gauge) and maximum deviation of 0.009% FS.

Apart from the required pressure measurement for calculation of the turbine performance, several other pressure sensors were installed on the volute circumference in order to monitor the pressure pulse propagation within the volute. All the pressure transducers are connected to individual signal conditioner as explained in steady flow pressure measurement section. The output of the conditioner-amplifier module was connected to the high-speed analogue-to-digital PCI card, NI 6034E by National Instruments. All the transducer-conditioner-amplifier output was referenced to a trigger pulse from the pulse generator via the shaft encoder in order to ensure that all pressure reading as well as the other recorded parameters was in similar time phase. The recording processes are done using in-house programmed LabView software.

Calibration. In order to establish proper transfer function for each transducer from voltage to gauge pressure, a calibrator unit Druck DPI 610 was utilized where a linear correlation was obtained. The maximum deviation of the calibrator is < 0.025% FS. For the conversion from static to absolute pressure, an atmospheric pressure during day of testing was specified.
2.3.1.3 Temperature

Steady flow condition. For steady condition, the temperature required for calculation of turbine performance involves only the temperature at the turbine inlet, that is, the measurement plane. However, a few more thermocouples are installed on the test rig for monitoring purposes. For instance, in order to keep the airflow at elevated and constant temperature to prevent condensation due to expansion in the turbine, thermocouples are also installed at the heater. Three different types of thermocouples were used during testing: E-type, K-type and T-type. In order to measure the temperature of a moving fluid, significant attention has to be given to the incompressibility effect. The influence of compressibility on Mach number causes the measured temperature to fall between the static \( T_s \) and total temperature \( T_0 \), therefore demanding the employment of a recovery factor. However, at low Mach number \( (< 0.3) \), Yahya (1982) has demonstrated that the compressibility effect is not significant, therefore the measurement does not have to be corrected. Therefore, since the thermocouples located upstream the orifice plate are subjected to low velocity airflow, the static temperature are measured directly without any recovery factor. On the other hand, at the measurement plane, the Mach number of the airflow has increased thus compressibility effect has to be considered. The recovery factor is defined in Equation \ref{eq:recovery_factor} and the corresponding corrected static pressure is given in Equation \ref{eq:corrected_static_pressure}

\[
r = \frac{T_{\text{meas}} - T_s}{T_0 - T_s} \tag{2.13}
\]

\[
T_s = \frac{T_{\text{meas}}}{1 + r \left( \frac{\gamma - 1}{2} \right) M^2} \tag{2.14}
\]

where \( r \) is the recovery factor of both thermocouples used at the measurement plane and \( T_{\text{meas}} \) is the original temperature reading of the sensor. The recovery factor was predetermined where the thermocouple is immersed in a wind-tunnel where the flow is accelerated gradually and the total temperature and pressure are measured at the plenum before the test section while the corresponding static values are measured.
at the test section.

**Pulsating Flow Condition.** The pulsating nature of airflow passing through the thermocouples requires evaluation of instantaneous static temperature by the isentropic relationship between the pressure and temperature. The instantaneous static temperature are evaluated by Equation 2.15.

\[
T_{s,\text{inst}} = T_{\text{mean}} \left( \frac{P_{s,\text{inst}}}{P_{\text{mean}}} \right)^{\left( \frac{\gamma-1}{\gamma} \right)}
\]  

(2.15)

where \(T_{s,\text{inst}}\) is the instantaneous static temperature, \(T_{\text{mean}}\) is the time-mean static temperature measured using the T-type thermocouple, \(P_{s,\text{inst}}\) is the instantaneous static pressure measured using high-response pressure transducer and \(P_{\text{mean}}\) is the time-mean pressure measured using static pressure tapping.

**Calibration.** The thermocouple calibration procedure is described in [Hakeem and Khezzar] (1994). All the thermocouples were calibrated at the freezing (273.15 K) and boiling point (373.15 K) of water with an additional room temperature point (293 K). Local pressure variations were taken into consideration in cases where it is necessary. A mercury bulb thermometer with a resolution of 0.1 K was used for the room temperature point calibration. Repeated calibration test shows temperature reading repeatability within ±0.4 K.

### 2.3.1.4 Rotational speed

Measurement of the rotational of the turbine is made using a reflective optical-switch of type Omron EE-SX4101 with integrated amplifier. The speed sensor, which is attached into the dynamometer, is an infra-red optical sensor which is triggered by a 10-toothed wheel mounted on the turbine shaft.

**Steady Flow Condition.** In steady flow testing, the 10 pulses per revolution (as a result of 10-toothed wheel) produced by the optical sensor are 'de-rated' to a single pulse. The single pulse is then used as a digital gate for a 16MHz clock in a 16-
bit counter. The time for one revolution of the turbine wheel is measured, and subsequently converted into DC voltage. The output is such that the increasing speed results in decreasing output voltage. The output of the counter is connected to the National Instrument FieldPoint analogue input module NI FP-AI-110, which is then recorded during the testing. Plot of the rotational speed in RPS versus voltage is illustrated in Figure 2.4.

![Figure 2.4: Turbine rotational speed as the function of the 16-bit counter input](Extracted from Rajoo, 2007)

Calibration. The transfer function relating turbine speed and output voltage as calibrated using a 5 kHz square wave signal generator as replacement to the optical sensor output that correspond to 500RPS turbine speed. The speed reading from the circuit shows the accuracy of $500 \pm 0.017$ RPS. To constantly check the calibration point, the turbine speed was directly monitored by FFT analysis of the vibration signal from the turbine. The first harmonic which corresponds directly to the turbine speed is tracked, in which for both cases the turbine speed match each other with consistent accuracy.

Pulsating Flow Condition. Since the actual instantaneous power depends heavily on instantaneous torque, thus the rotational acceleration, it is crucial to resolve the signal output from the optical sensor. For this purpose, the 10 pulses signal output was not de-rated but is kept as 10 pulses and is used as a gate for an internal reference 20MHz clock. Hence the time required for the turbine shaft to rotate for
Figure 2.5: Main components of the eddy current dynamometer (Extracted from Szymko, 2006)

Figure 2.6: Turbine rotational speed (Extracted from Szymko, 2006)
a given angle can be obtained and the angular acceleration can be evaluated. In
order to keep the measurement in phase with other parameters, output pulses from
the optical sensor was referenced to a trigger mark from the pulse generator via the
shaft encoder.

Calibration. For the pulsating condition, the calibration is required mainly to incor-
porate the irregularity in manufacturing of the tooth since the angular gap might
not be completely even for the whole circumference. For this reason, Szymko (2006)
recorded the 10 pulses output instantaneously while the turbine is rotating at steady
speed. This allowed a distinct patent of every 10th pulses to be seen as indicated in
Figure 2.6 Therefore, the dynamic angle \( \theta_n \) between each tooth could be calculated
based on the \( i \)-th segment of angular speed (\( \omega_i \)) and per-revolution averaged angular
speed (\( \omega_{mean} \)) as given in Equation 2.16.

\[
\theta_n = \frac{\omega_i}{\omega_{mean}} \frac{2\pi}{10}
\]  

(2.16)

The evaluation of dynamic angle \( \theta_n \) in Equation 2.16 has resulted in an improvement
of measurement uncertainty from \( \pm 7.85 \) RPS to \( \pm 0.16 \) RPS

2.3.1.5 Power measurement

As explained earlier in this chapter, the eddy current dynamometer is currently em-
ployed for measuring power generated by the turbine. This allows broader range
of operating condition to be evaluated as compared to the conventional compressor
as the loading device. The employment of eddy current dynamometer also offers
improvement to overcome inertia issue by the hydraulic dynamometer. The per-
manent magnet eddy current dynamometer used in this research was developed at
Imperial College by Szymko (2006). It provides braking to the turbine wheel using
principle of eddy-current by incorporating 14 ground Neodymium-Iron-Baron
magnets of 12mm depth onto a rotor. The arrangement is such that the magnet
rotates together with the turbine wheel while co-axially spins to a set of stationary
plates known as the stators. Two stators are used and are located at both sides
of the magnet (see Figure 2.5). These stator plates are heavily cooled to dissipate heat and their temperatures are consistently monitored throughout the experiment. Both stators are attached to the main body of the dynamometer which sits on a set of gimbal bearing held by a load cell coupling. Therefore, reaction of the torque that originated from the shaft that is transferred to the main body and subsequently the load cell could be measured directly.

Steady flow condition. In steady state condition, the torque ($\tau$) was measured directly from the dynamometer reaction exerted by the load cell. The torque is measured with a cantilever beam load cell of typeTedea Huntleigh 1040-I-20 and the output of the load cell is connected to Flyde FE-492-BBS Mini-Bal bridge conditioner and Flyde FE-351-UA Uni-Amp amplifier modules. The signal is then connected to National Instruments FieldPoint analogue input module NI FP-AI-110. With the value of rotational speed ($N$) available from optical speed sensor, the actual power of the turbine wheel is obtained by Equation 2.17

$$\dot{W}_{act} = 2\pi N \tau$$

(2.17)

Pulsating flow condition. Under pulsating flow condition, the instantaneous torque is required in order to calculate instantaneous turbine performance. This is achieved by considering the instantaneous torque, $\tau_{inst}$ as the summation of mean, $\tau_{mean}$ and fluctuating torque, $\tau_{fluc}$

$$\tau_{inst} = \tau_{mean} + \tau_{fluc}$$

(2.18)

The fluctuating torque is obtained by the product of the rotor angular acceleration ($\alpha$) with its polar moment of inertia ($I$) as given in Equation 2.19. On the other hand, the mean torque is obtained by using procedure described in steady flow condition.

$$\tau_{fluc} = I \cdot \alpha = I \cdot \frac{d\omega}{dt}$$

(2.19)
The first derivative of the angular speed in Equation 2.19 corresponds to the acceleration of the rotating component and it is calculated with the first central difference numerical technique.

**Calibration.** For obtaining the transfer function of the load cell torque to the voltage, static torque calibration was conducted. A calibration arm of known length (0.599 m) is attached radially on the periphery of the dynamometer with its loading parallel to the load cell. The water, oil and air supply pipe that is externally connected to the dynamometer that sits on gimbal bearing, therefore might result in preloading of the load cell. To compensate for this effect, the dynamometer is constantly supplied with water, oil and air running at operational level. Known weight of loading is then applied for the range of torque expected during testing and the transfer function is established. The calibration was also done on a regular basis during period of testing to ensure consistency. The combined calibration points suggests an uncertainty in the torque measurement of ±0.025Nm.

### 2.3.1.6 Miscellaneous

Polar moment of inertia. The rotating component polar moment of inertia was obtained using the tri-filar suspension method. The polar moment of inertia of the whole rotating assembly \( I \) is \( 4.563 \times 10^{-4} \text{ kgm}^2 \). Meanwhile the polar moment of inertia for the rotor wheel only is \( 9.858 \times 10^{-5} \text{ kgm}^2 \). The Root-Sum-Square RSS uncertainty in the calculation of polar moment of inertia is \( 1.55 \times 10^{-5} \text{ kgm}^2 \).

### 2.4 Uncertainty analysis

The parameters obtained from measurements explained in section 2.3 each consists of errors that caused by measurement instruments and its associated data acquisition modules. These errors will ultimately propagate and influence the final turbine performance parameters. Therefore, it is necessary to evaluate these uncertainty originated from individual parameters in order to obtain final uncertainty value.
from the turbine efficiency, mass flow parameter, pressure ratio and velocity ratio. The evaluation of the propagated uncertainties was made by performing the Root-Sum-Squares (RSS) of all the associated variable uncertainties, as shown in Equation 2.20. This particular methodology was proposed by Kline and McClintock (1953) and further developed by Moffat (1982). Stern et al. (1999) later gave a proper description of the uncertainty methodology based on the ASME and AIAA standards. The uncertainty analysis that is pertinent and specific to the current research was provided in detail by Szymko (2006). Szymko (2006) utilized the RSS method to perform the uncertainty analysis and has been used as the basis throughout the current work.

\[ \pm Par_{RSS} = \sqrt{\sum_{i=1}^{n} \left( \pm var_i \cdot \frac{\partial Par}{\partial var_i} \right)^2} \] (2.20)

where \( \pm Par_{RSS} \) is RSS uncertainty of parameter, \( \pm var_i \) is individual variable uncertainty and \( \frac{\partial Par}{\partial var_i} \) is the sensitivity coefficient.

Table 2.1: Sensitivity coefficient in the uncertainty analysis

<table>
<thead>
<tr>
<th></th>
<th>( \partial \dot{m} )</th>
<th>( \partial T_1 )</th>
<th>( \partial P_1 )</th>
<th>( \partial \tau )</th>
<th>( \partial N )</th>
<th>( \partial p_5 )</th>
</tr>
</thead>
<tbody>
<tr>
<td>( \partial \eta_{ts} )</td>
<td>44%</td>
<td>7%</td>
<td>15%</td>
<td>30%</td>
<td>0%</td>
<td>4%</td>
</tr>
<tr>
<td>( \partial VR )</td>
<td>28%</td>
<td>16%</td>
<td>45%</td>
<td>-</td>
<td>0%</td>
<td>11%</td>
</tr>
<tr>
<td>( \partial MFP )</td>
<td>72%</td>
<td>6%</td>
<td>21%</td>
<td>-</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>( \partial PR )</td>
<td>39%</td>
<td>3%</td>
<td>44%</td>
<td>-</td>
<td>-</td>
<td>13%</td>
</tr>
<tr>
<td>( \partial N_{eqv} )</td>
<td>21%</td>
<td>72%</td>
<td>6%</td>
<td>-</td>
<td>0%</td>
<td>-</td>
</tr>
</tbody>
</table>

The RSS uncertainty calculation for a turbine performance parameters (total-to-static efficiency, mass flow parameter, velocity ratio and pressure ratio and equivalent speed) are given in Equation 2.21 to 2.25. The table of sensitivity coefficient associated to each variable and the dependent parameter is shown in Table 2.1.
\[ \pm (\eta_{ts})_{RSS} = \sqrt{\left( \pm \dot{m} \cdot \frac{\partial \eta_{ts}}{\partial \dot{m}} \right)^2 + \left( \pm T_1 \cdot \frac{\partial \eta_{ts}}{\partial T_1} \right)^2 + \left( \pm P_1 \cdot \frac{\partial \eta_{ts}}{\partial P_1} \right)^2 + \left( \pm \tau \cdot \frac{\partial \eta_{ts}}{\partial \tau} \right)^2 + \left( \pm N \cdot \frac{\partial \eta_{ts}}{\partial N} \right)^2 + \left( \pm P_5 \cdot \frac{\partial \eta_{ts}}{\partial P_5} \right)^2} \] (2.21)

\[ \pm (MFP)_{RSS} = \sqrt{\left( \pm \dot{m} \cdot \frac{\partial MFP}{\partial \dot{m}} \right)^2 + \left( \pm T_1 \cdot \frac{\partial MFP}{\partial T_1} \right)^2 + \left( \pm P_1 \cdot \frac{\partial MFP}{\partial P_1} \right)^2} \] (2.22)

\[ \pm (PR)_{RSS} = \sqrt{\left( \pm \dot{m} \cdot \frac{\partial PR}{\partial \dot{m}} \right)^2 + \left( \pm T_1 \cdot \frac{\partial PR}{\partial T_1} \right)^2 + \left( \pm P_1 \cdot \frac{\partial PR}{\partial P_1} \right)^2 + \left( \pm P_5 \cdot \frac{\partial PR}{\partial P_5} \right)^2} \] (2.23)

\[ \pm (VR)_{RSS} = \sqrt{\left( \pm \dot{m} \cdot \frac{\partial VR}{\partial \dot{m}} \right)^2 + \left( \pm T_1 \cdot \frac{\partial VR}{\partial T_1} \right)^2 + \left( \pm P_1 \cdot \frac{\partial VR}{\partial P_1} \right)^2 + \left( \pm P_5 \cdot \frac{\partial VR}{\partial P_5} \right)^2} \] (2.24)

\[ \pm (N_{eqv})_{RSS} = \sqrt{\left( \pm \dot{m} \cdot \frac{\partial N_{eqv}}{\partial \dot{m}} \right)^2 + \left( \pm T_1 \cdot \frac{\partial N_{eqv}}{\partial T_1} \right)^2 + \left( \pm P_1 \cdot \frac{\partial N_{eqv}}{\partial P_1} \right)^2 + \left( \pm N \cdot \frac{\partial N_{eqv}}{\partial N} \right)^2} \] (2.25)

### 2.5 Data processing for unsteady measurement

As the unsteady flow measurements are collected at high sampling rate, the data obtained is subjected to external noise such as vibrations that is not related to the measured properties. For this purpose, ensemble averaging and filtering are used. These methods and instruments were originally developed by Rajoo (2007) and Szymko (2006).
Chapter 2. Experimental methodology

The measurements that are recorded for each unsteady parameter are taken for 50 pulses. With the traversing system that has 36 points, the total recorded data points are 1800 cycles. These points are ensemble averaged to a single cycle for the subsequent processing and analysis. For the hotwire reading, the 36 traversing points are integrated using British Standard method after the 50 cycles for each traversing point were individually ensemble averaged. The application of ensemble averaging procedure is useful to reduce the random noise and non cyclic variation in the acquired pulsating signal, where the signal-to-noise ratio can be improved by a factor of \( \sqrt{n} \).

Although the data have been ensemble averaged, some of the noises still remain. It is therefore necessary to filter the ensemble averaged data. Rajoo (2007) utilized a low pass Finite Impulse Response (FIR) filter to smooth the traces without losing its primary features. This filter is part of the built-in routine in the LabVIEW program. The filtering and ensemble averaging process produced one final data set of each unsteady flow property for a single pulse that is used for further analysis.

Another issue with unsteady data processing is the data acquired by the speed sensor. As the speed of the turbine wheel is detected by recording the time taken for the rotating assembly to move a nominally fixed angular distance (toothed gate), the sampling rate is not constant. This introduced a mismatched between data output from this particular sensor and the other data which was sampled at exactly the same rate of 20kHz. Therefore the analogue data re-sampling had to be undertaken. This process is done by fitting a cubic spline through each of the unequally spaced speed data points (Szymko, 2006).
Chapter 3

Simulation methodology

3.1 Introduction

This section intends to describe the development of computational model used in this research. Development of the geometry for computation domain and its meshing process are also described in details. Application of steady and unsteady boundary conditions on the domain inlet and exit, and steps taken to calculate the required parameters either on local computer or High Performance Computer (HPC) are also included.

3.2 Background of Computational Fluid Dynamics

Computational Fluid Dynamics (CFD) analyses are significantly employed in the current research to enhance understanding of the flow behaviour inside the turbine stage. In general, CFD is the art of solving partial differential flow equations, primarily the continuity equation and the momentum or Navier-Stokes equations.

The continuity equation is defined as
∂ρ \frac{\partial}{\partial t} + \nabla \cdot (\rho U) = 0 \quad (3.1)

For incompressible flow, this equation can be reduced to

\nabla \cdot U = 0 \quad (3.2)

Meanwhile, the momentum or Navier-Stokes equation is defined as

\frac{DU}{Dt} = -\frac{1}{\rho} \nabla p + \nu \nabla^2 U \quad (3.3)

Since the primary objective of simulations in the current research is to understand and analyse the flow field within the turbine stage, established commercial software was used rather than developing a new solver. For this purpose, commercial software Ansys CFX 14.1 has been chosen.

It is well known that the flow in most of the engineering applications is in the turbulent flow regime. This includes flow within the current turbocharger application. The existence of turbulent necessitates the explanation of how it might influence the flow equations. Reynolds in 1894 introduced the decomposition of the velocity $U(x,t)$ into its mean $\langle U(x,t) \rangle$ and its fluctuation $u(x,t)$ as described in Equation 3.4.

\[ u(x,t) = U(x,t) - \langle U(x,t) \rangle \quad (3.4) \]

This equation is known as Reynolds decomposition. The Reynolds decomposition ultimately results in modification of the original continuity and Navier-Stokes equations. It follows from the continuity equation 3.1 to equation 3.5

\nabla \cdot U = \nabla \cdot (\langle U \rangle + u) = 0 \quad (3.5) \]
By subtraction of mean velocity solenoidal (divergence free vector) condition,

$$\nabla \cdot \langle U \rangle = 0$$  \hspace{1cm} (3.6)

the resulting fluctuation result in equation 3.7

$$\nabla \cdot u = 0$$  \hspace{1cm} (3.7)

The integration of fluctuation velocity components to the original Navier-Stokes equation is less simple due to the existence of non-linear convective terms. The substantial derivative of velocity gives

$$\frac{DU_j}{Dt} = \frac{\partial U_j}{\partial t} + \frac{\partial}{\partial x_i} (U_i U_j)$$  \hspace{1cm} (3.8)

so that the mean is

$$\langle \frac{DU_j}{Dt} \rangle = \frac{\partial \langle U_j \rangle}{\partial t} + \frac{\partial}{\partial x_i} \langle (U_i U_j) \rangle$$  \hspace{1cm} (3.9)

By substituting the Reynolds decomposition, the nonlinear term becomes

$$\langle U_i U_j \rangle = \langle (\langle U_i \rangle + u_i)(\langle U_j \rangle + u_j) \rangle = \langle \langle U_i \rangle \langle U_j \rangle + u_i \langle U_j \rangle + u_j \langle U_i \rangle + u_i u_j \rangle = \langle U_i \rangle \langle U_j \rangle + \langle u_i u_j \rangle$$  \hspace{1cm} (3.10)

By substituting back into the mean substantial derivatives, the equation becomes

$$\langle \frac{DU_j}{Dt} \rangle = \frac{\partial \langle U_j \rangle}{\partial t} + \frac{\partial}{\partial x_i} \langle (U_i \langle U_j \rangle + \langle u_i u_j \rangle) \rangle$$  \hspace{1cm} (3.11)

Since $\frac{\partial \langle U_i \rangle}{\partial x_i} = 0$, therefore
\[
\langle \frac{D U_j}{D t} \rangle = \frac{\partial \langle U_j \rangle}{\partial t} + \langle U_i \rangle \frac{\partial \langle U_j \rangle}{\partial x_i} + \frac{\partial}{\partial x_i} \langle u_i u_j \rangle \tag{3.12}
\]

The first two terms on the right hand side of the equation \ref{3.12} can be simplified by defining the mean substantial derivative to be

\[
\frac{\bar{D}}{D t} = \frac{\partial}{\partial t} + \langle U \rangle \cdot \nabla \tag{3.13}
\]

Therefore, the equation becomes

\[
\langle \frac{D U_j}{D t} \rangle = \frac{\bar{D} \langle U_j \rangle}{D t} + \frac{\partial}{\partial x_i} \langle u_i u_j \rangle \tag{3.14}
\]

Finally, the mean momentum or Reynolds equation becomes

\[
\frac{\bar{D} \langle U_j \rangle}{D t} = \nu \nabla^2 \langle U_j \rangle - \frac{\partial \langle u_i u_j \rangle}{\partial x_i} - \frac{1}{\rho} \frac{\partial \langle p \rangle}{\partial x_j} \tag{3.15}
\]

The final equation that includes the fluctuation velocity components is called the Reynolds Averaged Navier-Stokes (RANS) equation. This equation contains all the parameters that exist in the original Navier-Stokes equation with the addition of the Reynolds stress terms (\(\langle u_i u_j \rangle\)). However, the additional terms impose a new problem that now the numbers of the unknowns exceed the number of available equations. This issue is commonly called as closure problem. This situation therefore requires an additional model, generally called the turbulent models in order to solve for additional stress terms before integrating the solution back into the original equation.

For the current work, the two-equation k-epsilon (\(k - \varepsilon\)) turbulent model has been chosen to solve the closure problem in the RANS equation. The main reason behind selection of this particular model is due to its ability to predict the turbocharger turbine performance with sufficient accuracy as described in Palfreyman (2004) and Newton (2014). The model is also known due to its wide applications in turbulence.
flow and is commercially available for most of the computational software, making it popular among CFD users.

The turbulent kinetic energy, \( k \) (defined as the variance of velocity fluctuations) and its dissipation rate, \( \varepsilon \), hence the name k-epsilon model come to light from the turbulent-viscosity hypothesis proposed by Boussinesq in 1887. This hypothesis states that the deviatoric Reynolds stress is proportional to the mean rate of strain by the factor of turbulent viscosity. This method is described in details by Pope (2000). With this hypothesis, it is possible to express the Reynolds stress term into effective viscosity which is the summation of molecular and turbulent viscosity. The effective viscosity is then integrated into the main RANS equation, and could be solved numerically without the closure problem.

In \( k-\varepsilon \) model, the transport equations for both turbulent quantities \( k \) and \( \varepsilon \) are solved. The definitions of both quantities are then used to calculate the turbulent viscosity as demonstrated in Equation 3.16

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

where \( C_\mu = 0.09 \) is a constant. This value could be used in the boundary layer but it is not sufficiently accurate at wall proximity \( (y^+ < 50) \). Therefore, modifications to the standard \( k-\varepsilon \) model are necessary for the viscous near-wall region. Several suggestions has been made over the years to include a damping factor in the calculation in order to reduce the over prediction problems (Pope, 2000).

The turbulent boundary layer close to the wall has been known to be substantially different from its laminar counterpart, thus requires additional efforts to obtain the wall condition, for example the wall shear stress. Empirical and Direct Numerical Simulation (DNS) data has shown that for turbulent boundary layer, there are multiple layer stages according to their positions from the wall where each of them behaves differently. The layer right next to the wall is known as the viscous sub layer where the fluid is dominantly influenced by molecular viscosity. The next layer after sub layer region is the logarithmic region where turbulence dominates.
the mixing process. Furthermore, the intermediary between viscous sub layer and the logarithmic layer is called the buffer layer.

In order to resolve this condition, in conjunction with previously mentioned $k-\varepsilon$ model, the ‘wall function’ approach is utilized to solve the steep profiles of velocity at near-wall region of turbulent boundary layer. The original idea of wall function methodology was developed in 1972 by Launder and Spalding where the turbulence models are not solved close to the wall. This means employing the empirical relationship to bridge the viscosity affected sublayer region for the mean flow and turbulence transport equations and connect the wall condition to variables at the node closest to the wall.

Furthermore, the available empirical relationship requires the closest node to the wall to be at the starting edge of logarithmic law region of the turbulence boundary layer where the relationship between wall shear stress and velocity is universally clear. In the log-law region the wall shear wall stress is related to the near wall tangential velocity through logarithmic relationship, hence the name log-law region. The relationship is described in equation (3.17)

$$u^+ = \frac{U_t}{u_r} = \frac{1}{K} \ln(y^+) + C$$

where

$$y^+ = \frac{\rho \Delta y u_r}{\mu}$$

and

$$u_r = \left( \frac{\tau_w}{\rho} \right)^{\frac{1}{2}}$$

where $u^+$ is the near wall velocity, $u_r$ is the friction velocity, $U_t$ is the known velocity tangent to the wall, $\Delta y$ is the distance from wall to the known tangential velocity,
$y^+$ is the dimensionless distance from the wall, $\tau_\omega$ is the wall shear stress and $K$ is the von Karman constant.

However, equation [3.17] has a problem of singularity as the velocity, $U_\tau$ approach zero due to separation. To overcome this problem, in the logarithmic region, an alternative velocity scale is used to represent $u^+$

$$u^* = C_\mu^{\frac{1}{4}} k^{\frac{1}{2}}$$

(3.20)

This scale has the useful property that it does not go to zero if $U_\tau$ goes to zero. Therefore, the following explicit equation for $u_\tau$ can be obtained

$$u_\tau = \frac{U_\tau}{K \ln(y^*) + C}$$

(3.21)

The absolute value of the wall shear stress $\tau_\omega$, is then obtained from

$$\tau_\omega = \rho u^* u_\tau$$

(3.22)

where

$$y^* = \frac{\rho \Delta y u^*}{\mu}$$

(3.23)

and $u^*$ is defined earlier.

Since the wall function approach only identify the log-law region as the basis of the relationship between tangential velocity close to the wall and wall shear stress, it is essential that the first node of near wall meshes fall into the log-law region. For instance, if the mesh is too fine and falls into the viscous sub layer, different empirical relationship of parameters on the node and wall would result in erroneous wall shear stress calculation. Therefore, refining the mesh even more close to the wall at this stage does not necessarily result in increasing prediction accuracy. Therefore, in
order to overcome this issue, there is a formulation developed by Ansys CFX called the Scalable Wall Function (Ansys, 2006). This feature come into use in order to ensure the closest node to the wall is at the starting point of log-law region from the wall which is approximately equivalent to $y^+ = 11.06$. Therefore all mesh points that are below this region will be ignored.

There are however several known limitations regarding the use of $k - \varepsilon$. Some of them is that this model is not suitable to be used for flow with aggressive boundary layer separation and flow with sudden change in the mean strain rate (Pope (2000), Ansys (2006)). Validation practice is therefore essential in order to ensure that the model is viable. For the current research, the validation procedures are explained in detail in the next following chapters.

### 3.3 Development of three-dimensional numerical model

In general, the 3-D model development starts by building the geometries, followed with the meshing process. Subsequently, the model is assembled together and boundary conditions are set at appropriate locations. Afterwards, the actual computing took place either using a local desktop for relatively small jobs (single passage simulations) or using High Performance Computer (HPC) for bigger jobs (full stage and unsteady simulations). Then, the simulations results are ready for scrutiny during post-processing phase.

In this particular research, the computational domain consists of three major sub-domain which are the volute, vanes and the rotor. Each of the geometry is built and meshed with different technique using different commercial packages where all of them are described in detail in the next section. In the current work, structured mesh elements are employed for vanes and rotor domain. For the volute, unstructured mesh elements are employed.
3.3.1 Geometry creation and meshing for volute, vanes and rotor

**Volute.** The original design of the turbine volute used in the current research is built by Rajoo (2007) as part of his PhD project. Rajoo (2007) modified the Holset H3B volute by adding a vane section in order to test for different types of vanes. Rajoo (2007) also fabricated and tested the volute and therefore provided the preliminary results for validations that are explained in subsequent chapters. The volute casing was modified in Solidworks commercial package, retaining its inside surface of cross-section in order to create the geometry of the volume within the volute stage. Additional 0.2 mm x 13.79 mm area was artificially added to the geometry at the tongue region to allow flow recirculation that existed in the actual volute.

Furthermore, in order to accommodate the pulsating flow simulations, the volute inlet is extended to allow the inlet boundary condition to be defined at the position equivalent to the measurement plane (refer Chapter 2). This extension is necessary to reflect the actual recorded pulsating data to be used as boundary condition, since the flow will travel at similar distance as in actual testing condition before entering the volute inlet. Moreover, this extension also allows the flow to developed and mixed before entering the volute inlet, therefore allowing the employment of the area averaged total pressure at the inlet of the domain. Development of the volute geometry as well as the extended inlet duct is shown in Figure 3.1.

![Figure 3.1: Volute and inlet duct geometry in Solidworks](image)
The meshing process for the volute domain is done using ICEM CFD commercial software. The software is capable of generating full hexahedral mesh with complete control capability of its arrangement from the user. This ensures flexibility that is very important while meshing a complex geometry such as the volute. Since there are multiple steps involved in mesh generation using ICEM CFD, the volute has been divided into five parts (including the inlet duct) to simplify the process. Details about the divided geometries are shown in Figure 3.2.

![Figure 3.2: The divided geometry of the volute](image)

All the boundaries such as the volute inlet, outlet, wall and interfaces between different parts of volute are defined during meshing process (see Figure 3.3). This procedure of specifying boundary surfaces is crucial in order to allow appropriate boundary conditions to be specified at the correct location prior to calculations. The resulting meshed components are then translated into \textit{.cfx5} files which are readable by Ansys CFX.

**Vanes.** Unlike the process in previous section that requires the volute to be built first before meshing, the development of vanes domain only requires the profile lines of the hub, shroud and the blade itself for a complete single passage mesh to be generated. For this purpose, a commercial software namely Ansys TurboGrid is used. Using this software, the geometry is supplied through definition of the abovementioned profile lines. The vanes that are utilized throughout the entire simulation are based on NACA 0015 profile with the lean geometry of 50\(^\circ\) from the hub surface. The vanes consist of 15 blades with different chord length of 22.3 mm and 26.3 mm for hub and shroud respectively. This is to ensure even spacing
Figure 3.3: Final meshing and assembly of the volute

of the interspace between the vanes trailing edge and the mixed flow rotor leading edge. These vanes were specifically designed to match 3 dimensional geometry of mixed flow turbine leading edge (Rajoo, 2007). It is worthwhile to note that the vane angle is defined based on the vane position at the shroud end between line connecting centre of the turbine wheel rotation and the pivot point of the vane and the vane camberline. The dimension of the nozzle vane employed in this research is visualized in Figure 3.4.

Another important issue during mesh development of the vanes is the tip clearance. The vanes were originally designed to have 0.15 mm clearance at both end walls. However, further investigations by taking a closer look at the manufactured volute indicate that the clearances on both sides are different. At the shroud side where the pivot arm is, the actual clearance of 0.2 mm is recorded. Meanwhile, at the hub side it was found that the clearance is negligible. However, since the addition of tip clearance could potentially prolonged the simulation period, single passage vane simulations were carried out to investigate three cases, one with no clearance at all (V1), one with ideal clearance of 0.15 mm on both sides (V2) and finally with actual
clearance value of 0.2 mm at the shroud wall (V3). Results of this investigation are shown in Figure 3.5.

From Figure 3.5 it can be clearly seen that there are significant differences of normalized static pressure trace between the blade surface without the tip clearance (V1) to that of with tip clearance (V2 and V3). The differences are however, marginal between case V2 and V3. From this investigation it can be concluded that under estimating the importance of tip clearance of the blade could result in misleading outcome of the simulation. To address particular case in this research, the vane domain with 0.2 mm clearance (V3) which also represented the actual working condition is selected to be used regardless of additional computational time require to calculate flow parameters at clearance region.

**Rotor.** Since Ansys Turbogrid requires the definition of the rotor blade profile lines as described above, considerable amount of time has been devoted into obtaining these lines. The task is less simple than generating profile lines of the vanes since the rotor blades has complex curvatures, thus require more profile lines. Therefore, 9 profile lines for a single blade are generated and this is done by employing Bezier Polynomials. The rotor that is currently used in this research was originally de-
Chapter 3. Simulation methodology

Figure 3.5: Comparison of normalized static pressure distribution at mid-span of the vane surfaces between CFD and experimental result with different vanes tip clearance (a) V1, (b) V2 and (c) V3

3.3. Development of three-dimensional numerical model
signed in-house by Abidat (1991) with the designated name ‘Rotor A’. Summary of Rotor A geometry is shown in Table 3.1. Furthermore, all the parameters for the Bezier Polynomials constants are extracted from Abidat (1991) and Palfreyman and Martinez-Botas (2005). These constants effectively serve as guides to the Bezier curve in order to define the rotor camberline, hub and shroud as seen in Figure 3.6. Since the curves generated by the polynomials were in polar coordinate form, they were then converted into Cartesian coordinate which can then be uploaded into Turbogrid. The construction of the blade profile lines are visualized in Figure 3.7.

Table 3.1: Geometrical feature of Rotor A

<table>
<thead>
<tr>
<th>Geometrical Feature</th>
<th>Dimension</th>
</tr>
</thead>
<tbody>
<tr>
<td>Leading Edge Tip Diameter [mm]</td>
<td>95.14</td>
</tr>
<tr>
<td>Leading Edge Span Height [mm]</td>
<td>18.00</td>
</tr>
<tr>
<td>Trailing Edge Tip Diameter [mm]</td>
<td>78.65</td>
</tr>
<tr>
<td>Trailing Edge Span Height [mm]</td>
<td>25.79</td>
</tr>
<tr>
<td>Cone Angle [°]</td>
<td>40.00</td>
</tr>
<tr>
<td>Leading Edge Blade Angle [°]</td>
<td>20.00</td>
</tr>
<tr>
<td>Root Mean Radius at Trailing Edge [mm]</td>
<td>52.00</td>
</tr>
<tr>
<td>Length of Axial Chord [mm]</td>
<td>40.00</td>
</tr>
<tr>
<td>Number of Blades</td>
<td>12</td>
</tr>
<tr>
<td>Tip Gap Height (% of blade span)</td>
<td>3</td>
</tr>
</tbody>
</table>

The mesh files of both vane and rotor blade are then exported into .gtm format. This is the format that is readable in Ansys CFX. All of the domains are assembled together in Ansys CFX Pre before the flow boundary conditions can be defined. Summary of the nodes distribution for all the meshed domains are included in Table 3.2.

### 3.3.2 Model assembly

As described in previous section, assembly of the meshed domains are executed in Ansys CFX Pre. This software is part of Ansys CFX package and its main
3.3. Development of three-dimensional numerical model

Figure 3.6: (a) Camberline and (b) Hub and shroud profile generated from Bezier Polynomial
Figure 3.7: 3-D representation of the generated rotor blade profiles

Table 3.2: Nodes distribution and type of mesh for all components

<table>
<thead>
<tr>
<th>Domain</th>
<th>Number of Nodes</th>
<th>Type of mesh</th>
</tr>
</thead>
<tbody>
<tr>
<td>Inlet Duct</td>
<td>242 320</td>
<td>Unstructured Hexahedral</td>
</tr>
<tr>
<td>Volute</td>
<td>730 016</td>
<td>Unstructured Hexahedral</td>
</tr>
<tr>
<td>Nozzle</td>
<td>590 160</td>
<td>Structured Hexahedral</td>
</tr>
<tr>
<td>Rotor</td>
<td>2 600 436</td>
<td>Structured Hexahedral</td>
</tr>
<tr>
<td>Total</td>
<td>4 162 932</td>
<td></td>
</tr>
</tbody>
</table>
purpose is to serve as platform to prepare the domain before moving on to computing stage. All the discretized geometries are assembled and the interfaces between the geometries are specified. Particular attention is given to the interface between the nozzle (stationary domain) and the rotor wheel (rotating domain). For this surface, the transient interface is chosen in simulation to allow rotation of the turbine wheel domain in relation to the stationary vanes. This method will result in a more realistic prediction of the flow field than that of 'frozen rotor' method as in Lam et al. (2002). This is because the transient effect is taken into account, although at the expense of significant increment of the computational time.

![Figure 3.8: Assembly of the domains in CFX-Pre](image)

### 3.3.3 Boundary conditions

At the inlet boundary of the domain (Duct Inlet), a time varying total pressure and total temperature is specified during pulsating flow simulations. The direction of inlet flow is defined so that the only velocity component that exists is normal to the inlet plane. The data for inlet boundary condition are taken directly from the experimentally acquired values at the measurement plane. One sample of boundary condition is shown in Figure 3.9 for 20 Hz pulsating flow at 50% speed. The outlet boundary condition requires the static pressure values. Therefore, constant atmospheric pressure is applied at the boundary plane. Moreover a no-slip boundary condition was specified at all walls including the vanes and the rotor blades.
Chapter 3. Simulation methodology

Figure 3.9: Total Pressure inlet boundary condition

Ansys CFX does not support the time dependent speed and acceleration of the rotating domain, thus a constant speed is set for the turbine wheel during simulation. The application of constant turbine speed during pulsating flow simulations remain valid since the maximum speed fluctuations during testing is found to be only within 1.5% of the cycle averaged value. Moreover, the time step is set to be equivalent to 1\(^{st}\) rotation of the turbine, or \(\Delta t = 5.556 \times 10^{-6}\) s and \(\Delta t = 3.472 \times 10^{-6}\) s for 50% (30000 rpm) and 80% (48000 rpm) wheel speed respectively. Similar time steps are applied for different pulsating frequency. This eventually results in the 20 Hz case to produce more data points than the 80 Hz case. In order to begin the pulsating flow simulation, an initial steady-state file is required to assist the convergence. For this purpose, the total inlet pressure for the initial file is set to be the mean value of the individual pulsating condition. For the steady-state simulations, similar boundary conditions are specified with the absence of time varying inlet total pressure and temperature.

3.4 Conclusion

The steps taken to create a full three-dimensional model of the turbine stage has been described. The effect of inaccurate representation of the blade tip gap are also
presented. Overall, the total number of over 4 million nodes have been created to represent the turbocharger turbine domain to be solved on the High Performance Computer (HPC) facility at Imperial College London. The average computing time for one pulsating flow simulation using 8 processors is 28 days. In total 8 cases of different speed and frequency were simulated. These simulations result in the massive requirement of memory storage of over 4Tb of data. Results derived from these simulations are presented in the subsequent chapters of this thesis.
Chapter 4

Steady flow turbine operation

4.1 Introduction

This section provides detailed flow field review of the turbine operation under steady flow conditions. The first part of this section discusses the validation procedures undertaken for the computational model against experimental data. Subsequently, steady flow characteristics of the flow field within the turbine stage are discussed. This analysis includes flow field visualisation of multiple reference planes relative to the turbine blade geometry. Also included in the analysis is the detailed flow angle analysis at different operation points which is later used as a comparisons with pulsating flow conditions in the next chapter.

4.2 Validation exercises

Since most of the discussion are based on the CFD results, extensive validation exercises are therefore necessary in order to ensure the model viability. In doing so, the parameters of turbine performance which are pressure ratio, velocity ratio, mass flow parameter and turbine efficiency which were obtained from CFD calculations are compared directly with the available experimental data.
Chapter 4. Steady flow turbine operation

Figure 4.1: Comparisons of Mass Flow Parameter between Experiment and CFD calculation

Figure 4.1 shows the plot of mass flow parameter against pressure ratio the turbine rotating at 50% design speed (30000 rpm). Both CFD and experimental data are plotted together on the same axis in order to see direct comparison. It can be seen that the developed full stage model used in this research are able to capture the experimental trend of mass flow parameter as well as its magnitude at any given pressure ratio. The Root Mean Square of the deviation over the entire simulation operation range is recorded to be 2%.

The model validation for turbine efficiency has been proven to be more difficult to achieve relative to the validation of mass flow parameter as demonstrated by previous researchers. This issue is attributed mainly due to the efficiency calculation itself which is derived from multiple parameters such as torque, temperature and pressure. Furthermore, the employment of fixed value of certain parameters such as specific heat in order to shorten the computational time could also lead to over or under prediction of turbine efficiency.

The comparison of steady flow total-to-static efficiency between CFD and experiment is plotted in Figure 4.2. It can be seen that CFD calculations under predicted the efficiency value at velocity ratio lower than 0.85%. Furthermore, maximum under prediction of efficiency is recorded to be 5 efficiency points at velocity ratio of
Chapter 4. Steady flow turbine operation

Figure 4.2: Comparison of total-to-static efficiency between Experiment and CFD calculation

0.36. The RMS deviation of CFD calculation from experimental data is 2 efficiency points. The calculated efficiency match the magnitude obtained in experiment at 0.85 velocity ratio. At velocity ratio higher than that, slight over prediction of the CFD model is recorded.

4.3 Results and discussion

This section presents the flow field behaviour within the turbocharger turbine stage starting from the volute, the vanes, and finally the rotor domain. For this purpose, a single case of 30000 rpm with pressure ratio of 1.3 (130 kPa inlet total pressure) was selected due to its ability to achieve peak performance for the particular speed. Therefore the analysis represents the flow field behaviour at optimum steady state operating condition. In the later stage of this section, another two steady flow cases is selected at higher and lower pressure ratio. The aims for selecting other operating points are to enable comparison of flow field between the optimum and off-design condition, and to investigate whether any of this condition is repeatable during unsteady flow operation.
Figure 4.3 shows the static pressure contour within the connecting duct and the volute at 50% span (see top right of Figure 4.3). There are also a few cross-section pressure contours (volute inlet, 0°, 90°, 180° and 270° circumferential location) to show the spatial variation of pressure distribution and progression at each section. It can be seen from Figure 4.3 that there is little pressure difference between the duct inlet and volute inlet relative to the overall pressure change within the stage and that at the volute inlet, the pressure is uniform albeit a small pressure drop is seen at the upper right corner of the volute inlet cross section. Downstream the volute inlet, it can be clearly seen that the pressure starting to drop as the flow turns and enters the vane domain. The pressure reduced gradually towards the vane inlet in relatively uniform manner except at the circumference end of the volute (340° – 360°). At this location the pressure drops significantly towards the volute tongue, therefore resulting in uneven recirculation flow as explained later in this chapter.

The velocity distribution in the inlet duct and volute stage is shown in Figure 4.4. In general, the flow within the volute stage accelerates circumferentially (due to reduction of radial distance of the volute centroid to the centre of rotation) and
Figure 4.4: Velocity distribution across the volute stage (PR = 1.3, 30000 rpm)

radially as it moves towards the vane inlet. As the vane is leaned and thus have shorter hub than its shroud, one would expect that the radial flow acceleration in the volute to be uneven between hub and shroud. However, the velocity contour at each cross section indicated that the flow accelerates almost radially. Therefore, this behaviour indicated that the flow distribution in the volute is not influenced by the blade geometry downstream but depended on how the volute itself was designed. Indeed the volute is the derivative of a Holset H3B volute which originally designed for a radial flow turbine. Moreover, Figure 4.4 also shows that close to the tongue, the velocity is not evenly distributed and accelerated more than the other section at similar radial location. Suhrmann et al. (2012) conducted extensive analysis with regards to the influence of the volute geometry to the overall flow distribution downstream the tongue. Suhrmann et al. (2012) indicated that a small tongue radius and angle could lead to better turbine efficiency but could also increase the probability of wheel blade high cycle fatigue failure. According to Suhrmann et al. (2012), a robust design would then require the tongue to have a larger tongue distance and angle. Despite the interesting observation relating the turbine performance with the volute tongue geometry, optimization of the tongue geometry is not part of the

4.3. Results and discussion

118
Chapter 4. Steady flow turbine operation

project described in this thesis and as such further discussion is not included.

Downstream the volute exit, the flow is expected to enter the vane domain at a flow angle of 69°. This value is selected during the design stage of the volute and is assumed to be constant throughout the exit of the volute circumference. However, this condition (i.e constant flow angle) is seldom met due to the free vortex assumption that is utilized during the design stage. With the CFD model developed for this research, it is possible to visualize this problem. Figure 4.6 shows the plot of flow angle distribution against the circumferential location of the volute. The location where the flow angle at the volute exit is measured is detailed in Figure 4.5. From Figure 4.6 it can be clearly seen that the flow angle does not maintain a constant value but it fluctuates due to the proximity of the vanes. Maximum fluctuation is recorded close to the tongue region with the magnitude of ±10°.

As the flow passes through the vane, the flow angle variation across the circumferential location has reduced significantly as compared to its variation at the volute exit/vane inlet. At the exit of the vanes, only small localized fluctuations are detected due to proximity to the turbine wheel inlet. There are no evidence of the wake flow from the vanes trailing edge could be seen at where the flow angle is measured. The magnitude of the mean flow angle is now 71° with only ±1° fluctuation. Therefore, this observation shows that current arrangement of vanes is sufficient to guide the flow and to minimize the substantial flow angle variations that occur at the volute exit. This ensures that the flow enters the rotor at a constant incidence angle which is very beneficial provided that the incidence angle falls into its optimum range. The relative and incidence angle are also plotted in the same axis in Figure 4.6. In the current operating condition, the incidence angle reads -10° which is close to the optimum limit, hence the maximum efficiency for the particular operating speed.

Although it seems like the existence of the guide vanes has a positive result in flow alignment, it does not guarantee high efficiency at off-design condition. The capability of the guide vanes to minimize the fluctuations of the flow angle across the rotor circumference also means that it is only efficiency if the incidence angle is kept

4.3. Results and discussion
at its optimum value. As the absolute flow angle changes at off-design condition, the incidence angle will also change and fall outside its optimum value at any circumferential location. Meanwhile, in the absence of the guide vanes where incidence angle fluctuates substantially, there are some portion of the rotor circumference that might be exposed to the optimum incidence at off-design condition. Therefore, the employment of vanes in a turbocharger turbine arrangement increases the maximum efficiency but only limited at the design operating condition. However, at off-design condition, the existence of the vanes row could result in low efficiency as the incidence angle fall away from its optimum value. The detailed explanations regarding the performance comparison of vaned and vaneless turbine arrangements are described in Chapter 6.

Even though the vanes are capable of improving the flow angle distributions at the volute exit circumferentially, the complexity of the turbocharger turbine geometry prevents similar distributions of the flow angle in spanwise direction. Figure 4.7 shows the plot of velocity components along the span of the rotor inlet at 180° circumferential location. From Figure 4.7, it can be seen that in general the flow has turned into its primary direction which is the streamwise direction. This is indicated by relatively high magnitude of the streamwise velocity as compared to the other velocity components. Along the span, it can also be seen that this velocity components varied as much as 65 m/s where the maximum value is recorded close to the shroud wall with the magnitude of 100 m/s. However, the magnitude of the streamwise velocity drops to 45 m/s at 0.85 span location. This feature is labelled

**Figure 4.5: Location of data measurement at mid-span of vane and rotor inlet**

**4.3. Results and discussion**
Chapter 4. Steady flow turbine operation

Figure 4.6: Flow angle at the vane inlet and the rotor inlet

X in the Figure (4.3). The sudden change in the velocity component over relatively short distance is due to blade tip clearance close to the shroud. The streamwise velocity then remains relatively constant at 50 m/s until 0.3 spanwise location. Closer to the hub, the velocity increases and peaks at 75 m/s before it drops again to 60 m/s at the hub proximity.

As for the spanwise velocity, a consistently negative value is recorded (see Figure 4.7). This feature indicated that there are fluid movement from hub to shroud of the rotor inlet, and therefore impeding the primary flow which is in the streamwise direction. This condition is worst at about 0.05 spanwise location close to the hub where the flow is moving from hub to shroud at 36 m/s. This behaviour occurs due to the geometry of the trailing edge of the vanes that has different radial distance from the centre of rotation in order to match the geometry of the mixed flow turbine. Therefore, the flow closer to the shroud turns from radial to axial direction at radial location higher than the hub. This in turns induces flow movement from hub to shroud at the leading edge of the rotor.

For the tangential velocity component, there is a strong change from a negative
value at shroud to a positive value at about 0.85 span location (labelled X in Figure 4.7). This behaviour is associated with the reduction of Coriolis acceleration that is acting on the fluid as it enters the rotating domain. Japikse and Baines (1994) indicated that the decreasing radius of the rotor causes uneven cross-passage force between the shroud and hub at the rotor. At negative incidence angle such as the current operating condition, this behaviour results in a secondary flow that is set up in the blade passage in the form of a circulation in the opposite direction to the passage rotation. For the current operating condition, the maximum difference of tangential velocity component as seen in Figure 4.7 is 85 m/s, a significant difference. Furthermore, close to the hub (less than 0.1 span location), the tangential velocity reduces to a relatively negligible value.

As the velocity components vary across the span of the rotor inlet, so does the flow angle. Figure 4.9 shows the plot of the absolute, relative and incidence flow angle across the spanwise location at 180° inlet of the rotor. It can be seen from Figure 4.9 that for the absolute flow angle, the value of 71° holds for most of the span length (from 0.22 to 0.9). However, close to the shroud wall, the flow angle reduces to 40°. In Figure 4.9, it is clearly seen that the incidence angle changes substantially.
Chapter 4. Steady flow turbine operation

Figure 4.8: Plot of total losses against the flow incidence angle at the rotor inlet

Figure 4.9: Flow angle components distribution across the span length of the rotor leading edge

4.3. Results and discussion
from hub to shroud. Close to the shroud, it has a very negative value of about -52°. Towards the hub, the incidence angle becomes less negative and turns to a positive value at about 0.4 span location. This flow angle levels-off at 0.2 span with the value of 5° and then reduces back to -25° at hub. The average value of the incidence angle at this location from hub to shroud is -9.8°. Despite the average value of the incidence angle of the current case which is outside the optimum limit indicated by Japikse and Baines (1994) (between -30° to -20°), the current case (-10° incidence angle) is still operating at its peak performance condition. From the CFD calculations at similar operating speed (30000 rpm), the optimum incidence (the angle which translates into minimum total losses) has a wider range than suggested by Japikse and Baines (1994) which is between -30° to 10°. This behaviour is visualized in Figure 4.8.

While analysing the flow field within the rotor stage, it is also useful to scrutinize the flow development by plotting the velocity and pressure contour on a few planes of specific orientations, namely streamwise, spanwise and blade-to-blade planes. Each orientation of the plane is represented in Figure 4.10. The employment of these planes would enable a more global view of the flow field behaviour bounded by the shroud wall, the hub, the suction surface as well as the pressure surface. The velocity and pressure contour of the streamwise planes is plotted side by side in Figure 4.11. It is also worth noting that the contour in Figure 4.11 is plotted such that an observer is facing towards the upstream incoming flow.

With regards to the velocity contour at the rotor leading edge, there exist a low velocity flow close to the pressure surface between 50% span and the shroud wall. As the flow moves further downstream to 50% streamwise location, the low velocity flow migrates closer to the hub (about 30% span) and also moved towards the mid-pitch region. Subsequently, at 75% streamwise location, the low velocity region has completely attached to the suction surface where it later moves towards the shroud wall as the flow reaches the rotor trailing edge.

Another interesting observation that can be seen in Figure 4.11 is the development of the tip leakage flow. At the leading edge as well as at 25% streamwise location,
one can hardly notice any development of the tip leakage flow. However, from 50% onwards, it can be clearly seen that the tip leakage flow has started to show its effect on the overall distribution of velocity in the particular plane as it emerged at the suction side near the shroud wall, and continues to exist until the trailing edge. The suppression of tip clearance effect on the velocity contour from the leading edge up to 25% streamwise location is attributed to the Coriolis effect as explained before. Since the tangential velocity at the shroud has a very negative value (-60 m/s opposite direction of the rotor blade) as seen in Figure 4.7, it is sufficient to prevent the flow from passing through the tip clearance from pressure side to suction side of the blade. However, as the flow gets further into the inducer, the flow loses its tangential component as it turns more axial, thus the Coriolis effect cease to exist. This allows the tip leakage flow to get through to the suction side and affect the entire flow structure of the particular plane. Downstream the 50% streamwise plane, the tip leakage flow mitigates further towards the centre pitch as it reaches the trailing edge. The clearance flow however, does not mitigate in spanwise direction therefore it always attaches to the shroud wall up until the flow reaches the trailing edge of the rotor.

Figure 4.10: Locations of pressure and velocity contour in the rotor stage for (a) streamwise plane, (b) spanwise plane and (c) blade-to-blade plane
The corresponding pressure contour for each streamwise plane is plotted on the right hand side of Figure 4.11. In general, for all streamwise plane except at trailing edge, it can be seen that the pressure gradually increase from suction to pressure surface. This behaviour is generally expected in any type of turbine. It is also noticed that there exist a very low pressure region at the suction surface of the rotor leading edge which is clear indication of flow separation despite optimum incidence flow angle. With regards to the tip leakage flow that has been discussed before, it can be seen that the influence begins to increase and inevitably has its effect on the pressure distribution starting at 50% streamwise location. The tip leakage flow that enters the turbine passage created a low pressure region as the flow velocity is high and potentially induces flow separation close to the suction surface. Therefore the low pressure region which is originally recorded close to the suction surface hub (25% streamwise) has migrated towards the shroud region as the flow proceeds downstream. Finally, as the flow exits the turbine, the pressure becomes stable and close to the ambient pressure where hardly any spatial variation in the rotor passage is noticed.

The velocity and pressure contour on three spanwise planes are plotted in Figure 4.12. The contour is plotted such that one is looking down towards the hub of the rotor. At 5% spanwise location which is very close to the hub (about 1 mm from the hub), it can be seen that there is a high velocity region attached close to the suction surface near inlet of the rotor. Generally, in a rotor passage, one would expect the flow velocity to increase as it reaches the rotor exit. While the statement holds true as the whole system is taken into consideration, it is however not the case for the flow field close the hub. There is even a low velocity island exists at about 75% streamwise location close to the pressure surface. The irregular observation as opposed to commonly accepted theory is mainly attributed to a combined effect of the flow turning very sharply close to the hub as well as the hub end of the wall shear stress.

At mid-span, a small separation region is developed at the pressure surface near the rotor inlet (labelled as X in Figure 4.12 at 50% span). The most severe and highly irregular flow field is recorded at 95% spanwise location. Two major separation
Figure 4.11: Velocity and pressure distribution at each streamwise location of the rotor
regions are detected. The first one occurs close to the turbine leading edge and up to 25% streamwise location near the pressure surface (labelled as Y in Figure 4.12). Meanwhile, the second and more severe occur at about 45% streamwise location close to the suction surface (labelled as Z in Figure 4.12). The second separation developed further downstream where it occupies 55% of the pitch and therefore disturbing the primary flow. This behaviour is related to the proximity of the 95% spanwise plane to the shroud wall and as such, the tip clearance. It has been discussed before in Figure 4.11 that the tip leakage flow starts to develop at about 50% streamwise location where its effect to the rest of the passage can be clearly seen in Figure 4.12. The pressure contour at 95% spanwise location also shows the effect of having tip leakage flow to the rotor blade capability to generate torque. It can be seen that at any similar streamwise location, the pressure would gradually increase as the pitch distance from suction surface increases. However, close to the pressure surface, the pressure suddenly drops back (circled in Figure 4.12), therefore minimizing the blade capability to generate torque. This phenomenon occurs due to the flow ability at particular spanwise location to accelerate and slip through the tip clearance region, therefore reducing the pressure close to pressure surface.

In order to assess the power generation capability of the blade at different spanwise position, the pressure loading on a blade located at 180° circumferential location is plotted. This plot can be seen in Figure 4.13 where three lines of different spanwise locations which are at 5%, 50% and 95%. The static pressure in this plot is normalized with the turbine inlet total pressure to simplify the analysis. The plot in Figure 4.13 represents the pressure loading exerted by the particular blade at any streamwise location. It is desirable to have uniform pressure loading at all streamwise and spanwise locations of the blade. Figure 4.13 indicated that the loading exerted at mid-span has the most steady distribution from the turbine blade leading edge (streamwise location = 0.0) to its trailing edge (streamwise location = 1.0). At 95% span, there exist a large pressure difference between pressure and suction surface at 0.53 streamwise location and then the difference rapidly becomes smaller toward the blade trailing edge. This behaviour is a direct result of the flow separation induced by strong turning curvature as well as the tip leakage flow as previously explained. The lowest blade loading amongst all spanwise location is seen at 5% location where
Figure 4.12: Velocity and pressure distribution at each spanwise location of the rotor
Figure 4.13: Pressure loading on the blade at optimum operating condition of 30000 rpm

the pressure loading downstream of 0.6 streamwise location reduced quickly.

To complete the full 3-dimensional description of the flow field within the rotor passage, the velocity and pressure contour at blade-to-blade locations is also plotted. Figure 4.14 shows 3 locations of blade-to-blade planes on which the velocity and pressure contour is plotted, namely 5% blade-to-blade (close to pressure surface of a passage), 50% blade-to-blade (mid-passage) and 95% blade-to-blade (close to suction surface of a passage). The plane locations are shown in Figure 4.10. In general, the pressure contour indicated that at similar streamwise location, the pressure is always higher at shroud as compared to hub. This is due to reduced passage width as the flow gets closer to the hub, thus forcing the flow to accelerate and in turns, reduce the pressure. Furthermore, by comparing the flow structure using the velocity contour in Figure 4.14, it can be seen that as the blade-to-blade plane moves away from the pressure surface, there is a low velocity region that mitigates from about 20% streamwise close to the shroud to 50% streamwise and about 30 spanwise. Subsequently, this low velocity region in the passage moves to 65% streamwise close to the hub as the blade-to-blade plane approach the suction surface. Furthermore, the effect of tip leakage is clearly visible in the velocity and pressure contours at 95% blade-to-blade. Another obvious observation in the pressure contour of Figure
Figure 4.14: Velocity and Pressure distribution at 5%, 50% and 95% blade-to-blade location
is that there is a flow separation exists at the rotor hub. This observation is clear at 50% and 95% blade-to-blade locations and but does not appear on 5% plane. The particular feature of the flow separation near the rotor hub is commonly seen in mixed flow turbine as indicated by Palfreyman and Martinez-Botas (2002). Meanwhile, for the radial turbine, such feature ceases to exist (Palfreyman and Martinez-Botas 2002).

As the 3-dimensional flow structure has been described in the optimum condition case, it is therefore necessary for the comparison to be made with off-design condition. The comparisons mainly aim to investigate the effect of different operating condition on the flow angle (specifically incidence angle) distribution at the rotor inlet. It can be seen earlier in Figure 4.8 that the distribution of incidence angle as the operating condition move away from the optimum either to a higher or lower pressure ratio is not symmetrical. The increment of incidence angle as the pressure ratio increases away from the optimum condition seems to be slower than the its decrement as the pressure decreases. Therefore, one would expect that the vaned volute will be beneficial at high pressure ratio and as such would result in higher efficiency in unsteady flow condition. However, the averaged incidence angle alone cannot be used as an ultimate tool toward overall turbocharger turbine performance. Indeed, the whole 3-dimensional effect of the flow angle distribution has to be taken into consideration. For this comparison purpose, two arbitrary off-design conditions were chosen at higher and lower pressure ratio than the optimum condition.

Figure 4.15 shows the absolute angle variation at mid span of the entire rotor inlet circumference for different operating conditions. It can be seen that for different operating condition, the existence of vanes has turned the flow to an average value of 71°. However, as the absolute flow angle is resolved across the rotor inlet circumference as indicated in Figure 4.15, it becomes clear that the flow is not constant, and behaves differently as the pressure ratio increases. It can be seen that the increment in pressure ratio results in high fluctuation range of the flow angle (6° at 2.2 PR compared to 1° at 1.17 PR).

Figure 4.16 shows the incidence angle distribution throughout the circumference of
Figure 4.15: Plot of the absolute flow angle for $360^\circ$ circumferential location at the rotor inlet for different operating conditions.

Figure 4.16: Plot of the incidence flow angle for $360^\circ$ circumferential location at the rotor inlet for different operating conditions.
rotor inlet for different cases. It can be seen that the deviation of incidence angle from its averaged value for every case also increases as the pressure ratio increased. This feature is expected since the increment of swallowing capacity would result in higher meridional velocity and as such, the relative flow angle\textsuperscript{1}. It is also seen that higher range of fluctuation at high pressure ratio that exists in the absolute flow angle plot (Figure 4.15) also exists in the incidence angle plot.

Figure 4.17 shows the incidence angle plot of the incidence angle across the spanwise location for all operating conditions. It can be seen in Figure 4.17 that the low pressure ratio case has the most evenly distributed incidence angle as compared to the other two cases. It also does not have sudden changes of incidence angle close to the shroud wall. This shows that the low momentum flow is easily guided by the vanes. However, close to the hub where the flow has started to turn into the axial direction, the reduction of flow angle between 0.2 spanwise location and the hub wall is clearly visible for all 3 cases.

Figure 4.17: Incidence angle at various spanwise positions for different operating conditions

\textsuperscript{1}the full feature of this behaviour is described in detail in Chapter 5
4.4 Conclusion

From the full three-dimensional steady state analysis of the flow passage as well as the comparison made at different off-design operating condition, it is clear that the vane stage have a positive impact on the turbine performance at its design condition. However, their existence have different effect on the flow at higher and lower pressure ratio than the optimum condition (off-design) even though the average output of the flow angle is identical. At high pressure ratio, the high momentum flow is forced to turn in a short distance. This results in uneven distribution of flow angle across the inlet circumference of the rotor. Meanwhile, during the low pressure ratio operations, the vanes in current arrangement is more than capable to turn the flow to its intended absolute flow angle with little variation in pitchwise and spanwise direction. Therefore, low efficiency obtained during this operating condition is attributed mostly due to the mismatch of the incidence angle of the flow that enters the rotor. This investigation also discovers that a simple averaging procedure of a certain critical parameters, in this analysis the flow and incidence angles, could lead to misinterpretation of the predicted outcome of the rotor capability to perform efficiently.

The discussion so far focused on the steady state turbine operating conditions. In an actual engine, the introduction of pulsating flow as the result of reciprocating motion of the exhaust valves inevitably changes the flow features that are seen in this chapter. These features, made visually available by means of validated numerical calculations, are the focus of the subsequent chapter.
Chapter 5

Pulsating flow turbine operation

5.1 Introduction

This section provides detailed flow field review of the turbine operation under pulsating flow conditions. The first part of this section discusses the validation procedures, followed by the analysis. The two main analysis conducted in this chapter is the flow field comparison between steady and pulsating flow conditions, and also comparison of flow field between different pulsating flow frequencies.

5.2 Validation exercises

For pulsating flow turbine operation conditions, multiple CFD calculations were conducted at 50% and 80% design speed for each frequency (20, 40, 60 and 80 Hz). These simulations are very extensive and computationally expensive where more than 4Tb of post-processing data were gathered for total 8 simulations. All pulsating conditions were validated with available experimental results. To demonstrate the validation exercises undertaken for these simulations, a single turbine operating condition of 50% speed at 20 Hz is selected. Unlike the validation procedures for steady state turbine operating condition where the chosen validation parameters
are the derived quantities, the validation for pulsating turbine condition is focused towards more fundamental parameters. In this case, two parameters namely static pressure at 180° volute circumference and the turbine rotor torque have been selected. The selection of fundamental parameters instead of the turbine performance parameters is done in order to validate the parameters obtained by CFD to the parameters that are measured during the experimental works. This ensures that any deviations between CFD and experiment can be seen in time domain and a potential source of error could be determined.

Figure 5.1: Comparison of static pressure at 180° volute circumference between experiment and CFD calculation

Figure 5.1 shows the plot of static pressure at the centroid 180° volute circumference for both experiment as well as CFD calculation. It can be seen that CFD calculation is able to pick up the pressure trace well during the pressure increment and decrement instance, as well as the trough region of the pulse. In addition the calculated static pressure range also matched with the experimental data. However, at the pulse peak, CFD calculation shows some deviation in terms of the phase of the peak pressure. The peak pressure as indicated by CFD occurs about 10° phase angle later than that of experimental plot. Nevertheless, the magnitude of both peaks in good agreement to each other.

Following the validation of pressure trace in the volute stage, another parameter
Chapter 5. Pulsating flow turbine operation

Figure 5.2: Comparison of rotor torque between experiment and CFD calculation which is directly related to the turbine wheel, namely torque is plotted in Figure 5.2. Overall, the CFD calculation agrees well with experimental data even though the torque is calculated using different methods as explained in Chapter 2 and Chapter 3. The shifting of peak position that is seen previously in Figure 5.1 can still be seen in the torque trace while the magnitude of peak torque is still in agreement to each other. The validation of both parameters, despite slight shifting in the peak region still indicate that the developed model is able to capture the time dependant pulsating flow within the turbocharger turbine stage. Therefore, the viability of the model for pulsating flow environment is confirmed and is used for further discussion for the rest of this chapter.

5.3 Phase shift assessment

One of the interesting debates amongst researchers as explained in literature review section in Section 1.2.4 is the phase-shifting methodology. In this section, this methodology is explained in detail in terms of its importance in obtaining accurate instantaneous turbine performance characteristic as well as comparison between the two popular phase shifting techniques. The assessment in this chapter will allow a
correct and consistent phase-shifting method to be selected and subsequently used throughout the discussion.

The need for phase-shift is due to the different measurement location of the parameters required to evaluate the turbine instantaneous performance parameter. For instance, in order to obtain the turbine efficiency, the parameters needed are the isentropic and actual power. In the current research, the isentropic power are obtained at the measurement plane upstream the volute inlet, whereas the actual power are obtained at the rotor (see Figure 5.4). Therefore, in pulsating flow environment, the finite time required for both pressure and mass to propagate from the measurement plane to the rotor needs to be taken into account during calculation of the turbine instantaneous efficiency. Otherwise, the calculated instantaneous efficiency does not correspond to the exact instance of when the available energy is being converted to the rotor actual power. This ultimately results in not only abnormal but also misleading efficiency value.

The effect of shifted phase during data acquisition of isentropic and actual power is clearly demonstrated in Figure 5.3 which shows the instantaneous isentropic and actual power of the turbine for 50% speed at 20 Hz and 80 Hz without any phase-shifting method applied. A black dotted line plotted on the similar axis for both Figure 5.3 is the corresponding instantaneous turbine efficiency. It is obvious in Figure 5.3 that the actual and isentropic power is not in the same phase where actual power trend shifted to the right due to the measurement delay of the traveling pressure waves. Furthermore, it can also be seen from the efficiency plot that the value is far greater than unity especially in Figure 5.3(b) where the reason is ultimately due to inconsistencies in the phase on which the efficiency is evaluated. This is a clear indication that phase-shifting is needed in order to obtain accurate instantaneous turbine performance characteristics.

At the moment, there are already a few methods of phase-shifting techniques introduced by different researchers as detailed in Section 1.2.4. However there is also a more simplistic method to correct for the phase different is to evaluate the time phase by matching the peak point of isentropic and actual power. The other promising
Figure 5.3: Isentropic and actual power plot as well as its instantaneous efficiency without phase-shifting at 50% speed for (a) 20 Hz and (b) 80 Hz
phase-shifting approach is by using the summation of sonic and bulk flow velocity which has been used by Szymko (2006) and Rajoo (2007). In this section, these two methods of phase-shifting are evaluated and one of them will be used throughout the future analysis.

In order to evaluate both methods, evaluation of isentropic power is made at three different planes, namely the measurement plane, the vane inlet and the rotor inlet as indicated in Figure 5.4. These evaluations are made possible by employing the available validated CFD results. The instantaneous isentropic powers available at different planes are area averaged and serve as a comparison basis for calculation of instantaneous efficiency. At these planes, the magnitude of calculated instantaneous efficiency should therefore be highest when evaluated at rotor inlet, followed by vane inlet and finally at the measurement plane. These efficiency values should be in the constant order ($\eta_{\text{Rotor Inlet}} > \eta_{\text{Vane Inlet}} > \eta_{\text{Measurement Plane}}$) throughout a whole pulse cycle if the phase-shifting is conducted properly.

Figure 5.4: Isentropic power measurement locations

Figure 5.5(a) shows the calculated isentropic and actual power at different locations without any phase shifting. It is worthwhile to note that the plot of actual and isentropic power at the measurement location is similar to that of Figure 5.3(a). The right hand side of Figure 5.5(a) shows the calculated instantaneous efficiency evaluated at different location. It can be seen that not only the efficiency value is too high, but also it is also not arranged according to the correct order. If the
Figure 5.5: Total-to-static efficiency evaluated at different locations (a) without phase shifting, (b) phase shifted using peak power matching and (c) phase shifted using summation of bulk flow and sonic velocity
Chapter 5. Pulsating flow turbine operation

Phase-shifting procedure is done using the peak power matching (Figure 5.5(b)), the value and order of instantaneous efficiency at different locations improved greatly as indicated in the right hand side of Figure 5.5(b). However, there are still a few conditions which are out of order where the efficiency evaluated at the measurement plane exceeds that of rotor inlet. This irregularities are labelled X and Y in Figure 5.5(b). For the second method, where the phase-shifting is done using the summation of bulk and sonic velocity, the instantaneous efficiency plot improved even more as can be seen in Figure 5.5(c). Although, the irregularities of efficiency order at different location still exist.

There are several drawbacks that can potentially be deduced from both techniques. Use of first method (peak power matching) can only be employed when the peak of isentropic and actual power is clearly visible. Therefore as the pulsating frequency increases, the interferences originated from pressure wave reflection and superposition could result in difficulties in locating this peak. This example of difficulties can be clearly seen in Figure 5.3(b) where at the frequency of 80 Hz, no obvious peak can be selected. For the second method (summation of bulk flow and sonic velocity), in order to be able to calculate the time shift, additional information is needed which is the location of entry point at the turbine inlet. At the moment, this point is assumed to be at 180° from the volute tongue. The assumption is based on the information from previous researches as well as evidence found in the current research where phase-shifting using this location as entry point yielded reasonable efficiency values. Furthermore, a quick calculation using peak power matching Figure 5.5(b) revealed that the location of this entry point is 230°. However, this value seems not to be consistent throughout all tested frequencies. Analysis of the weakness of these two methods result in a definitive conclusion that phase-shifting using summation of bulk and sonic velocity is able to provide better results with less ambiguity to apply and calculate. Therefore, this method is used throughout the entire analysis involving instantaneous efficiency parameter.

It is the main intention over decades of turbocharger research to improve the device’s capability to recover as much energy as possible from the exhaust gas. With regards to automotive internal combustion engine which produce highly pulsating flow, this
means high instantaneous efficiency is demanded throughout the pulses. However, steady-state turbine performance analysis already indicated that efficiency always varies over operation range of a turbine. The introduction of pulsating flow condition has been proven to have altered the turbine performance relative to steady condition, and also worsen its efficiency. The inability to increase the efficiency of the turbine under pulsating flow condition is due to the lack of understanding on its behaviour during such condition. Therefore, the discussion in this section are focused on the flow field behaviour within the turbine stage under pulsating flow conditions. Due to the high computational demand to investigate this issue, not much literature is available, thus signifying the novelty of the current work.

![Figure 5.6: Instantaneous efficiency, isentropic power and actual power for different frequencies at 50% turbine speed](image)

Figure 5.6 shows the instantaneous isentropic and actual power as well as the instantaneous efficiency on the secondary axis for turbine operating at 50% design speed for variation of frequencies (20, 40, 60 and 80 Hz). In general, it can be seen from the Figure 5.6 that the turbine fails to maintain constantly high efficiency throughout a pulse cycle regardless of its operating frequency. As the frequency increases, the amplitude of isentropic and actual power varies more significantly with additional
secondary and tertiary peaks due to reflection and superposition of pressure wave. These features inevitably result in more unpredictable progression of instantaneous efficiency. Furthermore, there are also instances where the actual power exceeds isentropic power which results in instantaneous efficiency higher than unity, even after phase-shifting process is done. This behaviour was addressed before by previous researchers which attributed this observation due to free-wheeling of the turbine rotor at low pressure ratio instances as a result of its inertia.

In pulsating flow environment, even though maintaining constant efficiency is not possible, it is desirable to have high instantaneous efficiency at the instances where most of the power are available (peak isentropic power), i.e. high pressure ratio instance. In Figure 5.6, it can be seen that the desirable condition has not been met. In fact, the efficiency is low at high isentropic power instances. In the 20 Hz and 40 Hz cases, this behaviour occurs at phase angle between $50^0$ to $100^0$. Meanwhile for 40 Hz and 60 Hz, it occurs at phase angle between $30^0$ to $120^0$.

### 5.4 Incidence angle effect on the turbine performance

Unlike the ICE that is able to maintain efficient performance over relatively large operating range, turbo machines have narrow operating range and highly dependent on the flow angle (Watson and Janota, 1982). As such, the investigation is focused on the various flow angle associated with the turbine wheel.

In a pulsating flow condition, the changing nature of flow field with regards to time at the turbine inlet is best represented by incidence angle. Figure 5.7 shows the circumferentially averaged incidence angle progression over pulse phase angle for variation of pulsating frequencies. Optimum incidence range between $-20^0$ and $-30^0$ as indicated by Japikse and Baines (1994) is marked by a grey region in Figure 5.7. Since the optimum incidence should result in less separation and secondary flow in the turbine passages, it would be beneficial to design a turbine that could maintain...
this incidence throughout a pulse. However, this is proven difficult as incidence angle varies up to $\pm 50^0$ away from optimum value as indicated in Figure 5.7. High positive incidence angle value would result in separation at the blade suction surface whereas high negative incidence would induce separation at the blade pressure surface. As the consequence of both conditions, the effective flow area in the passage is reduced and therefore restricting primary flow.

With regards to the relationship of instantaneous incidence to the pressure ratio, it can be seen that the incidence angle tend to be at its highest point (highly positive) at maximum pressure ratio. This is a direct contradiction of the attempt to manipulate the incidence angle to be at its optimum value at this instance. This behaviour of incidence angle shifting can be explained by studying the velocity triangle in Figure 5.8.

In Figure 5.8, the notation 1 (black lines) and 2 (green lines) are representative of two different rotor inlet flow conditions. The total pressure at 2 is higher than total pressure at 1. As the pressure increase from condition 1 to 2, the absolute flow angle, $\alpha$, remains as the volute was designed to provide a constant absolute flow angle (in this case $69^0$). With the mass flow rate increases from condition 1 to 2, the
meridional component of absolute flow velocity also increases as $\dot{m} = \rho C_m A^1$. This reaction resulted in the change of direction of the relative flow angle from opposite the turbine rotation ($W_1$) to the same direction as the turbine rotation ($W_2$). This prompted the relative flow angle and incidence angle to increase. Therefore, from this analysis, it can be seen that at a constant turbine speed, the value of incidence angle tend to be high at high pressure ratio instances in a pulse, which is not a favourable situation. However, as one would expect, it is possible to introduce changes to the incidence angle by changing the turbine speed.

Figure 5.9 shows the plot of instantaneous incidence angle for various flow frequency at 80% turbine speed. It can be seen that the maximum incidence angle is reduced as compared to its values at 50% speed (Figure 5.7). For instance, at 20 Hz frequency, the maximum incidence at 80% speed is recorded to be at $8^0$ as compared to $28^0$ at 50% speed. This also means that the maximum incidence angle has been brought closer to the optimum values. This development resulted in the increment of the cycle-averaged efficiency tabulated in Table 5.1.

Figure 5.10 shows the plot of instantaneous efficiency altogether with the plot of actual and isentropic power at 80% turbine speed. Although it is still difficult to characterize the trend of the instantaneous efficiency plot, considerable improvement

---

1It can be shown that at low Mach number the changes of mass flow rate is governed mostly by changes of velocity than density (assuming constant flow area)
Chapter 5. Pulsating flow turbine operation

Figure 5.9: Instantaneous incidence angle at 80% speed

Table 5.1: Cycle-averaged efficiency magnitude for all test conditions

<table>
<thead>
<tr>
<th>Frequency (Hz)</th>
<th>$\eta_{30000 \text{rpm}}$ (%)</th>
<th>$\eta_{48000 \text{rpm}}$ (%)</th>
<th>$\Delta \eta_{\eta_{48000 \text{rpm}} - \eta_{30000 \text{rpm}}}$ (%)</th>
</tr>
</thead>
<tbody>
<tr>
<td>40</td>
<td>62</td>
<td>70</td>
<td>+8</td>
</tr>
<tr>
<td>60</td>
<td>65</td>
<td>71</td>
<td>+6</td>
</tr>
<tr>
<td>80</td>
<td>63</td>
<td>72</td>
<td>+9</td>
</tr>
</tbody>
</table>

5.4. Incidence angle effect on the turbine performance
Figure 5.10: Instantaneous efficiency, isentropic power and actual power for different frequencies at 80% turbine speed
Chapter 5. Pulsating flow turbine operation

at high power instances is recorded. For example, at 20 Hz, the instantaneous efficiency at the particular instance for 80% speed is recorded to be 65%, 15% higher than the value recorded at equivalent condition for 50% speed (Figure 5.6). However, as the whole range of incidence angle is brought to a lower magnitude at higher speed, the incidence angle at low pressure ratio has a very high negative value. This translates into the efficiency penalty at low power instances as demonstrated in 20 Hz frequency in Figure 5.10 where the turbine has only 40% efficiency at the beginning and end of the pulse as compared to 60% at 50% speed. Nevertheless, the trough region of a pulse does not contributed much towards the average power output of the turbine. The focus to harness more available energy at high powered instances is also the reason why the power weighted-averaged for the turbine unsteady performance parameter is more suitable quantity than a time-averaged values. Additionally, power weighted-averaged efficiency also eliminate the phasing effect.

Due to the difficulty maintaining constant optimum incidence at all time, it is desirable to match the optimum incidence at high power instances in a pulse. However, the incidence angle tends to be at its maximum during this instance. Increase in speed that brought the incidence angle closer to its optimum value has resulted in the cycle-averaged efficiency increment, albeit the turbine suffers poor performance at the trough region of the pulse. Design of rotor inlet that incorporates more negative blade angle to match the optimum incidence could also help in improving the cycle-averaged turbine efficiency.

5.5   Pulsating flow vs. steady flow analysis

5.5.1   Flow angle comparison

In order to analyse the unsteady flow field during pulsating operations of the turbine, several conditions were selected on which the analyses are focused on. It would also be useful to compare those conditions to that of the equivalent steady state operating conditions. For this purpose, one with equivalent pressure ratio is selected. For the
Chapter 5. Pulsating flow turbine operation

particular analysis, a single operating condition of 50% turbine speed operating at 20 Hz pulsating frequency is selected. This point is chosen due to its pulse shape that is relatively free of any pressure superposition or reflection where only one clear peak is visible. Three pressure instances are selected with an additional point from steady state calculation. The instance of selected conditions are detailed and visualized in Table 5.2 and Figure 5.11 respectively.

Table 5.2: Unsteady and steady conditions for flow field analysis

<table>
<thead>
<tr>
<th>Point</th>
<th>Total Inlet Pressure [kPa]</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>pt1</td>
<td>160.0</td>
<td>increasing pressure</td>
</tr>
<tr>
<td>pt2</td>
<td>160.0</td>
<td>decreasing pressure</td>
</tr>
<tr>
<td>pt3</td>
<td>225.1</td>
<td>peak pressure</td>
</tr>
<tr>
<td>pt4</td>
<td>160.0</td>
<td>steady pressure</td>
</tr>
</tbody>
</table>

Pt1 and pt2 are taken at similar pressure ratio but at different instances whereas Pt3 is taken at the highest pressure ratio condition in the current case. The magnitude for pt1 and pt2 are selected at mid range pressure between the lowest and the highest pressure in the particular operating condition (50% speed at 20 Hz frequency). Meanwhile, pt4 is a condition of the steady state operation equivalent to Pt1 and Pt2.
Figure 5.12 shows the flow angle at the circumference of the vanes inlet for all four conditions. It is interesting to note that all four conditions indicated almost similar flow angle variation circumferentially. In general it can be seen that all four conditions indicated that the volute is capable of maintaining the flow angle at the average value of 73° from 0° until 200° circumferential position after which the flow angle drops gradually for about 10° before reaching the end of volute circumference. The minimum value of flow angle is recorded for pt1 at 350° with the magnitude of 60°. Meanwhile, large deviation towards high magnitude of flow angle occurs close to the tongue region (in the vicinity of 0° and 360°). This behaviour is seen for all four conditions. This indicated that the flow close to the tongue is almost tangential regardless of the inlet conditions whether it is steady or pulsating flow.

It has been indicated earlier in Figure 4.6 during steady state operation analysis that the vanes are responsible to provide uniformly distributed absolute flow angle throughout the inlet circumference of the rotor. However, at the proximity of the rotor blade, the flow angle experienced changes due to the effect of back pressure. In an optimum steady flow operations, these changes are recorded to be less than ±2° from the mean value. For the analysis in this section, the absolute flow angle distribution for four conditions described in Table 5.2 are plotted.
Figure 5.13 shows the plot of circumferential distribution of absolute flow angle at the rotor inlet for all four operating conditions. It can be seen in that the although the average value of the flow angle at these conditions are relatively similar (72°), there is a clear difference between the magnitude of flow angle deviation between steady (pt4) and unsteady conditions (pt1, pt2 and pt3) as the flow approaches the rotor inlet. Pt4 shows flow angle fluctuations in the range of 5° as compared to the other three unsteady conditions which have almost 20° range, an increment of 400%. This magnitude of fluctuation is even more than the fluctuation during high pressure steady state condition indicated earlier in Figure 4.15. It is also seen that the fluctuation range of the flow angle under pulsating flow condition is similar regardless of its pressure level. It is therefore clear from Figure 5.13 that the introduction of pulsating flow has resulted in unfavourable condition at the rotor inlet even though with the help of guide vanes.

The relative angle at the circumferential location of the rotor inlet for all four conditions is plotted in Figure 5.14. As all the conditions that are selected are at higher than optimum pressure ratio (1.3 during steady state condition), one would expect that the relative flow angle is positive in magnitude. In general, it can be seen in Figure 5.14 that the relative flow angle distribution for pt1, pt2 and pt3 are not as close together as their absolute flow angle distribution. It can be seen that pt4 still
maintain small range of fluctuations of the relative flow angle(8°). Pt1 indicated the most severe fluctuation amongst as compared to the other conditions. The range of relative flow angle fluctuation is up to 60° at this condition. Figure 5.14 also indicated that the initial expectation that pt1 and pt2 would operate closely due to similar pressure level at inlet is incorrect. In fact, the relative flow angle at pt2 is larger at most of the circumferential location than pt1. Smaller fluctuation range is recorded at pt2 as compared to pt1, indicating better circumferential distribution of the relative flow angle. The comparison with equivalent steady state condition (pt4) indicated that the relative flow angle during pressure decrement is likely to be similar to the steady state than during the pressure increment period. Additionally, during the pressure increment period (pt1), it can be seen that the tongue has a clear effect on relative flow angle where its value dropped very low as the flow approaches 360° circumferential location. This feature ceases to exist at any other chosen conditions. This shows that the pulsating flow field has more influence in deteriorating the flow angle during pressure increment than during pressure decrement period.

Figure 5.15 shows the plot of rotor efficiency against area-averaged incidence angle for a single case of pulsating condition (50% speed and 20 Hz frequency) as well for the steady state operations at multiple pressure ratio operating conditions. The black dots on Figure 5.15 marked the locations of interest as previously detailed in

5.5. Pulsating flow vs. steady flow analysis
It can be seen in Figure 5.15 that the hysteresis loop of the incidence angle is formed for pulsating flow condition where at any point of similar incidence, there are two different efficiency values. It is also clear from the Figure 5.15 that the turbine performance behaviour during pressure decrement period is closer to the steady state operation than during pressure increment period. This plot also serves as an indication that the operating condition of a turbine wheel under pulsating flow condition does not necessarily be similar at different instances even though the averaged incidence angle that enters the rotor passage is identical.

### 5.5.2 Flow field comparison

Figure 5.16 shows the pressure distribution downstream of the vanes row at peak pressure condition at 50% speed and 20 Hz frequency. The pressure contour range is plotted at mid-span with the turbine exit facing outward of the paper. There is a clear pressure variation at the vane inlet close to the tongue before the flow at the volute end recirculates back into the main flow. This feature is labelled as X in Figure 5.16. This flow distortion is already indicated earlier in Figure 5.12 where the flow...
angle is recorded to be as high as 85°, which is almost tangential. Furthermore, it is also seen in Figure 5.16 that the pressure variation at the particular region directly influenced the flow downstream the vane passage. Within the rotor passages, it can be seen that the pressure pattern is not repeating for each passage where there exist some variations of pressure between passages that have not been seen during steady state operations. However, no particular trend is seen with regards to these variations.

In order to visualize the details of the flow field within the passage, the pressure and velocity contour are plotted for a few spanwise planes for all four conditions of interest. Three spanwise planes at 5%, 50% and 95% spanwise location are selected and the positions of these planes are visualized in Figure 4.10.

Figure 5.17 shows the pressure contour plot for pt1, pt2 and pt4 at different blade-to-blade planes. It is clear from Figure 5.17 that the distribution of pressure field is significantly different from each other, albeit similar total pressure at the vanes inlet. The pressure field distribution for both pt1 and pt2 appear to be less uniform as compared to pt4 at all spanwise planes. It is also evidence from Figure 5.17 that the
Figure 5.17: Static pressure distribution on blade-to-blade planes for pt1, pt2 and pt4
low pressure (flow separation) region close to the suction surface within the passage is larger during the pressure decrement period (pt2) than the pressure increment (pt1) period during a pulse. The worst distribution of pressure field occur at pt1 and pt2 at 95% blade-to-blade plane due to the combined effect of tip leakage flow as well as the unsteadiness of the incoming flow. Even though there is a large low pressure region near the suction surface at 95% blade-to-blade plane for the steady condition (pt4), this region is limited only close to the suction surface until the flow reaches the rotor trailing edge. On the other hand, other conditions especially pt2 indicated that the low pressure region keeps growing and occupies most of the passage as the flow exit the turbine.

Figure 5.18 shows the velocity contour for pt1, pt2 and pt4 at different blade-to-blade plane. Since the selected inlet total pressure for the comparison of these three conditions are at higher-than-optimum pressure, one would expect that the velocity field distribution within the rotor passage is not as uniform as seen earlier in Figure 4.14. This behaviour occurs due to positive incidence angle as well as its non-uniformity at the rotor inlet. Despite that, it can still obvious in Figure 5.18 that the velocity distribution at pt1 and pt2 are more irregular than its distribution at pt4. As expected from the analysis of pressure contour in Figure 5.17 the velocity distribution of pt1 (increasing pressure) is rather different than pt2 (decreasing pressure).

Another phenomenon that can be seen in Figure 5.17 is the comparison of the velocity contour behaviour at 5% spanwise location between steady(pt4) and unsteady(pt1 and pt2) conditions. At pt4, it can be seen that the low velocity region emerged at about 30% up to 60% streamwise location close to the suction surface whereas for the other two unsteady conditions (pt1 and pt2), the low velocity region emerged close to the leading edge of the rotor up to about 50% streamwise location, also close to the suction surface.

The second part of the flow field comparison involves the three unsteady conditions which are pt1, pt2 and pt3. The pressure contours for these conditions are plotted in Figure 5.19. It can be seen that during pulsating flow conditions, the pressure
Figure 5.18: Velocity distribution on blade-to-blade planes for pt1, pt2 and pt4
distribution is distorted at all times especially close to the shroud wall (95% span) due to the reason explained before. At high pressure ratio (pt3), the region of low pressure at 5% spanwise location seems to be occupying half the passage pitch at 20% streamwise but also bounded only up to about 40% streamwise location close to the suction surface. Another interesting observation is that the low pressure region that exists in pt3 which is mostly close to the suction surface is very close in terms of its magnitude to the low pressure region recorded in pt1 and pt2.

Figure 5.19: Static pressure distribution on blade-to-blade planes for pt1, pt3 and pt2
Figure 5.20: Velocity distribution on blade-to-blade planes for pt1, pt3 and pt2
Figure 5.20 shows the velocity contour at different spanwise planes for pt1, pt2, and pt3. At 5% spanwise location, it can be seen that there is a low velocity region close to the suction surface and originated from the leading edge for all the conditions. This is one of the key differences in the velocity profile between pulsating and steady turbine operations where the low velocity occurs slightly downstream during steady operations as indicated in Figure 5.18. This behaviour is still visible at 50% spanwise location. However, this particular feature at mid-span is not unique to pulsating condition since it also exists during steady operations. Meanwhile, close to the shroud wall, the low velocity region ceases to exist close to the suction surface leading edge but has mitigated downstream for pt3. For pt1 and pt2, the low velocity now occurred close to the leading edge pressure surface.

5.6 Pulsating flow at different frequencies

By using the CFD data of the turbine under pulsating operations, it is also possible to investigate the flow field at similar level of pressure for each individual frequency. In this exercise, it is felt that the comparison at similar inlet total pressure level is more reasonable than comparing the flow field at peak pressure pulse for each frequency. Not only that the peak pressure occurs at different level of pressure which would compromise the ability to plot consistent contour, but also for some cases of higher frequency flow have multiple peaks due to pressure reflection and superposition in the turbine stage. Furthermore, this comparison would provide some information whether or not the flow patterns are repeatable at any frequency. As the pressure amplitude is different for each frequency, it is not possible to deduce a constant intermediate pressure level to conduct the analysis. Therefore, the pressure ratio of 1.4 is selected since it is the highest pressure ratio that could be achieve without the interference of multiple peaks and troughs particularly at 80 Hz flow frequency. Figure 5.21 shows the selected conditions for analysis for all flow frequencies.

Figure 5.22 shows the plot of flow angle at the volute exit/vane inlet against the
Chapter 5. Pulsating flow turbine operation

Figure 5.21: conditions of interest for the analysis at similar pressure level at different pulsating frequencies

Figure 5.22: Flow angle at vane inlet for all frequencies during the pressure increment instances

5.6. Pulsating flow at different frequencies
circumferential locations for all frequencies during the pressure increment period. It can be seen in Figure 5.22 that there are two groups of flow angle that are close to each other. 20 Hz and 40 Hz cases form one group while 60 Hz and 80 Hz cases form the other flow angle group which has 5° higher flow angle. However, towards the end of the volute (circumferential location greater than 270°), these two groups of flow angle merge to a similar fluctuating value until the end of the volute circumference. Close to the tongue region, the flow angle for all frequencies elevated to almost 100°. This feature occur after the flow angle dropped significantly at 330° volute circumference to 55° flow angle, 14° less than its intended angle. The flow angle that is larger than 90° (at 360° circumference) at the vane inlet indicated negative streamwise velocity which is a direct result from flow recirculation that causes the flow close to the tongue region to move away from the vane inlet. This behaviour potentially results in highly distorted flow field as discussed earlier in Figure 5.16, albeit at a different pressure level.

Figure 5.23: Absolute flow angle at rotor inlet for all frequencies during the pressure increment instances

Figure 5.23 shows the absolute flow angle plot against the circumferential location at the mid-span of rotor inlet for all flow frequencies during the pressure increment period. As the flow passes the vane passages, Figure 5.23 indicated that the groups of flow angle that existed at the vane inlet has now disappeared. The absolute flow angles at every frequency are now close to each other albeit still fluctuating.

5.6. Pulsating flow at different frequencies
due to proximity of the rotor blade. Despite that, not all flow angles fluctuate at similar phase to each other for the circumferential location less than $100^0$. For the particular circumferential location ($<100^0$), only the flow angle at 20 Hz and 40 Hz pulsating frequency fluctuate at similar phase whereas the other two frequencies has a rather different flow angle pattern. However, at higher than $100^0$ circumferential location, the flow angles for all frequencies are close to each other. At the end of the circumferential location, the flow angle at 40 Hz indicated higher value ($74^0$) than the other frequencies ($67^0$).

![Figure 5.24: Relative flow angle at rotor inlet for all frequencies during the pressure increment instances](image)

Figure 5.24: Relative flow angle at rotor inlet for all frequencies during the pressure increment instances

Another type of flow angle that is of interest in the analysis is the relative flow angle at rotor inlet due to its direct relationship with incidence angle ($i = \beta - 20^0$). Figure 5.24 shows the relative flow angle distribution for all flow frequency at the rotor inlet during the pressure increment instance. In general, Figure 5.24 indicated that between $0^0$ to $200^0$ circumferential location, there are two groups of relative flow angle at different level of magnitude. Downstream this location, it can be seen that these groups of flow angle merge with each other until the volute end. In terms of the magnitude of relative flow angle at circumferential location less than $200^0$, the 20 Hz case has the highest relative flow angle, followed by 40 Hz, 80 Hz and finally 60 Hz case. As for the two groups mentioned earlier, one at lower level of magnitude

5.6. Pulsating flow at different frequencies
Chapter 5. Pulsating flow turbine operation

consists of 60 Hz and 80 Hz flow frequencies where the other group consists of the other two flow frequencies. The reason for these separations of relative flow angle magnitude at different frequencies has yet to be determined. At higher than 200\(^0\) circumferential location, the trend of the magnitude of relative flow angle according to each flow frequencies cannot be determined properly as the order keep changing as circumferential location increases. Furthermore, as the two groups of flow angle merge with each other, the relative flow angle for 60 Hz and 80 Hz frequencies have increased at 200\(^0\) circumferential locations and then decreased again at 330\(^0\) circumference. On the other hand, the relative flow angle for 20 Hz and 40 Hz frequencies are relatively stable before suddenly decreased at 330\(^0\) circumferential location. In addition, the plot of relative flow angle in Figure 5.24 also shows that there is no direct relationship between the absolute and relative flow angle in terms of their trends across the rotor inlet circumference.

![Plot of flow angle at vane inlet for all frequencies during the pressure decrement instances](image)

Figure 5.25: Flow angle at vane inlet for all frequencies during the pressure decrement instances

Figure 5.25 shows the plot of flow angle against the circumferential location at mid-span of the vane inlet for all the flow frequencies during the pressure decrement period in a pulse. As opposed to Figure 5.22 (similar plot but during pressure increment period), Figure 5.25 shows that the flow angle for all frequencies are very close to each other except when the flow approaches the volute end. It can
be seen in Figure 5.25 that the flow angle for all frequencies recorded an average value of 72°. Furthermore there is no indication that the flow angle goes beyond 90° at this instance. This observation indicated that there is no negative streamwise flow (reverse flow) occurs close to the recirculation region albeit the flow angle is relatively high close to the tongue. This is a significant improvement of the flow angle distribution at this area when compared to the similar pressure level during pressure increment period (see Figure 5.22) where negative streamwise flow is observed. In Figure 5.25, the differences of flow angle according to individual frequency only emerged after the flow passes 270° circumferential location where the largest flow angle deviation from each other is recorded at 347° circumferential location with the magnitude of only 6°.

Figure 5.26: Absolute flow angle at rotor inlet for all frequencies during the pressure decrement instances

As the flow angle prior to the vanes during pressure decrement is more uniform than pressure increment period, one would expect that the flow angle at the vane exit will share similar attributes. Figure 5.26 shows the plot of absolute flow angle at the rotor inlet against the circumferential location for all flow frequencies during the pressure decrement period. In general, it can be seen that the vanes generally amplifies the local fluctuations caused by the blades, although maintaining similar averaged absolute flow angle for all flow frequencies. The seemingly out of phase

5.6. Pulsating flow at different frequencies
between 20 Hz/60 Hz and 40 Hz/80 Hz is attributed to the rotor blades location during the instance where the data is obtained. Another interesting observation that could be made from Figure 5.26 is that the amplitudes of the fluctuation of absolute flow angle for all frequencies are constant throughout the circumference of the rotor inlet. This behaviour is not seen earlier during pressure increment period (see Figure 5.23) where the flow angle fluctuations are higher at certain circumferential location that then others.

Figure 5.27: Relative flow angle at rotor inlet for all frequencies during the pressure decrement instances

Perhaps, the most significant difference between the flow angle during pressure increment and decrement period is the relative flow angle. Figure 5.27 shows the plot of relative flow angle at mid-span of rotor inlet against the circumferential location for different flow frequencies during pressure decrement period. From this figure, it can be seen that the local fluctuations of the relative flow angle are contained within a certain range. This means that the averaged incidence angle for all flow frequencies is identical. Moreover, this observation shows direct contradictions from the similar plot of relative flow angle during pressure increment period where the fluctuation span is a lot larger and strongly dependent on the individual frequency. The average relative flow angle during the particular pressure decrement instance is 30°, which corresponds to 10° incidence angle. Although it is fluctuating, the
relative flow angles maintain its positive value for all flow frequencies.

The analysis from this section revealed that the flow field could be behaving similarly or differently for each frequency depending on the instance. When the pressure is increasing, the flow angle distribution at the vane inlet is divided into two groups of 20 Hz/40 Hz and 60 Hz/80 Hz where the latter group indicated higher flow angle. These groups later disappear as the flow passes the vane rows. However, the plot of relative flow angle at similar location again revealed these groups, thus eliminate linkage between absolute and relative flow angle in terms of their trends across the rotor inlet circumference. Furthermore, the flow angle distribution improves significantly during pressure decrement period. At this instance, the range of localized fluctuations of absolute flow angle for all frequencies are constant throughout the circumference of the rotor inlet. The plot of relative flow angle during pressure decrement period also suggests that the localized fluctuation is contained within a certain range. These analyses support the findings in section 5.5.1 which indicates that the turbine performance during pressure decrement period is closer to steady state performance as compared to the pressure increment period. This appears to be true for all flow frequencies.

5.7 Flow ‘unsteadiness’ in the turbine stage

5.7.1 Review on Quasi-steady Mass Flow Parameter

The assumption that the turbine rotor would behave quasi-steadily due to short distance of the travelling flow as compared to the overall turbine stage (volute, vanes and rotor) is widely accepted. This section intends to evaluate this assumption by using the available CFD results.

Figure 5.28 shows the turbine swallowing capacity evaluated at different planes, namely volute inlet and rotor inlet for steady and pulsating operations. It is obvious from the plot that the swallowing capacity plots for every frequency fall into a
Figure 5.28: Mass flow parameter plot at (a) volute inlet and (b) rotor inlet

5.7. Flow ‘unsteadiness’ in the turbine stage
single line when they are evaluated at the rotor inlet, therefore proving the quasi-steady assumption. Not only that, the mass flow parameter obtained for pulsating flow operations coincide with the steady state mass flow parameter evaluated at the similar plane (rotor inlet). However, the comparison with the steady state swallowing capacity evaluated at the volute inlet indicated that the mass flow parameter evaluated at the rotor inlet has higher magnitude at all operating conditions. This presents a new issue since the quasi-steady assumption of the turbine deviates away from the conventional turbine map which is evaluated at the volute inlet. Therefore, this analysis revealed that even though the quasi-steady assumption for the rotor wheel is true, there is a need to adjust for the swallowing capacity differences that occur particularly at high pressure ratio conditions.

5.7.2 Lambda Parameter

In order to quantify the significance of unsteady flow condition to the turbine performance characteristics, several researchers such as Szymko et al. (2005) and Costall (2007) utilized the pressure modified Strouhal number (sometimes called reduced frequency) as the main parameter. The Strouhal number is defined as:

\[
St = \frac{fL_0}{U_0} = \frac{T_0}{t_0}
\]  

Greitzer et al. (2004) described the physical interpretation of the Strouhal number. If the fluid particle travel at velocity \( U_0 \) in the domain of length \( L_0 \), the Strouhal number is defined as the ratio of the time for fluid particle transport through the device \( \frac{L_0}{U_0} \) to the time scale associated with unsteadiness, in this case \( \frac{1}{f} \). Therefore, small values of \( St \) mean that fluid particles barely experience any change due to unsteadiness, while large values mean that fluid particle experience substantial variation during its transport time.

According to Equation 5.1, if the domain size and fluid particle velocity is kept the same, the value of Strouhal number will continue to increase if frequency increases.
This would mean that the influence unsteady effect will continue to dominate if frequency is increased. However, this condition is not necessarily true. As an example, the turbulent disturbance has very high frequency (in the order of 1MHz) but could be treated as quasi-steady due to its small amplitude. This ambiguity has become an important motivation to include the effect of pulse amplitude to properly characterize the level of ‘unsteadiness’ in the turbocharger turbine stage.

In addressing this issue, Greitzer et al. (2004) pointed out that one of the important definition of steady-state flow condition is that the system must not have any mass imbalance. Copeland et al. (2012) and Newton (2014) pointed out that the ratio between time-averaged mass flow changes in the domain and time-averaged mass flow through the domain could be used as an indication of mass imbalance in the turbine stage. This ratio is represented as

\[
\frac{2\Delta \rho V_0}{t_0} \div \rho_0 U_0 A_0 = \frac{2\Delta \rho}{\rho_0} \frac{V_0}{U_0 A_0 t_0} = \frac{2\Delta \rho}{\rho_0} \frac{L_0}{U_0 t_0} = \frac{2\Delta P}{\gamma P_0} = \Pi X St = \Lambda \quad (5.2)
\]

It is obvious that the result of this division could be represented as a product of the Strouhal number, St. Assuming an adiabatic system, the density terms in the equation could be interchanged with pressure. Therefore, the final mass imbalance ratio can be expressed as a product of the Strouhal number St and a pressure amplitude weighting factor given the symbol of capital Pi, Π.

\[
\frac{2\Delta \rho}{\rho_0} \frac{L_0}{U_0 t_0} = \frac{2\Delta \rho}{\rho_0} St = \frac{2\Delta P}{\gamma P_0} = \Pi X St = \Lambda \quad (5.3)
\]

Copeland et al. (2012) presented the final term of this derivation procedures as lambda parameter, Λ. If this parameter approaches unity, the average rate of mass change within the domain is of similar or comparable magnitude as the average rate of mass travelling in and out of the domain. Therefore there is a significant discrepancy between the mass flowing in and out of the domain and as such the system cannot be assumed to be quasi-steady.

Figure 5.29 shows the plot of three non-dimensional parameters which are Strouhal,
Figure 5.29: Strouhal, Pi and Lambda parameter plot against the flow frequency for both 50% and 80% turbine speed

Pi and Lambda parameter against the flow frequencies for 50% and 80% turbine speed. It can be seen that for the Strouhal number plot, the values continue to increase as the frequency increases. Constant offset of the Strouhal number between 50% and 80% turbine speed is also seen due to higher flow velocity required to spin the turbine at higher speed, thus lowering the Strouhal number. As for the plot of the Π parameter in Figure 5.29, it is clear that its values decrease as the frequency increases. This behaviour agrees with the definition of the Π parameter which is effectively the amplitude of the pressure pulse. However, the decrement of the Π value as the frequency increases is not as linear as the increment of the Strouhal number. The value of Π reduces much quicker at low frequency and then reduces rather slowly at higher frequency where it is expected to level off at certain minimum value. Both operations at 50% and 80% turbine speeds show marginal difference of Π value throughout the operating frequencies.

As indicated in Equation 5.3, the result of the multiplication of the Strouhal and Pi number is known as the Lambda parameter(Λ) which is also plotted in Figure 5.29. The plot of Λ indicated that the mass imbalance in the turbocharger turbine stage does not necessarily increase as the frequency increases. The highest mass imbalance
is recorded somewhere close to pulsating frequency of 60 Hz – 70 Hz (marked X in Figure 5.29). At higher frequency, the \( \Lambda \) value started to decrease. However, since the Strouhal number increased linearly as the frequency increased, and as the \( \Pi \) number reaches its minimum asymptotic value, the \( \Lambda \) value is expected to increase even more when the flow frequency is sufficiently high.

5.8 Pressure distribution on the turbine blade surface

This section intends to discover the power generation capability at particular locations of the turbine blades at different operating conditions. This is done using similar operating conditions indicated earlier in Table 5.2. Figure 5.30 shows the plot of normalized static pressure at different span of the blade surface for different operating conditions.

Figure 5.30 reveals that the most distorted pressure loading occur close to the hub region (Figure 5.30(a)) where multiple flow separations\(^1\) are detected as the flow travels the entire streamwise length. For current conditions where the relative flow angle is positive, the flow separation is likely to occur on the suction surface than the pressure surface. During pulsating flow turbine operations (pt1, pt2 and pt3), close to the hub region, the pressure difference between pressure and suction surface (therefore the blade power generation capability), is reduced to almost negligible value, even slightly negative value (work is transferred from the rotor blade to the fluid) between 50% to 60% streamwise location. This feature is indicated by label X in Figure 5.30. Meanwhile, for pt4 (steady-state condition), even though the separation region is still visible, the entire streamwise length of the blade is shows higher pressure level on the pressure surface than the suction surface.

At 50% spanwise location, it is interesting to see that pt1, pt2, and pt4 have almost similar pressure profile at the pressure surface. On the other hand, at the suction

\(^1\)indicated by sudden increase in \( \frac{P^p}{P_0^s} \) value
Figure 5.30: Plot of normalized surface pressure of the blade at (a) 5% span, (b) 50% span and (c) 95% span
surface, pt1 and pt2 have lower pressure distribution at all locations as compared to pt4. This observation suggests that at similar inlet total pressure, turbine operating under pulsating flow condition is capable of generating more power at mid-span of the blade as compared to its steady counterpart. Moreover, the power generated when the pressure is decreasing in a pulse (pt2) is higher at the blade mid-span as compared to the instance where the pressure is increasing (pt1). Meanwhile, at the highest pressure instance in a pulse (pt3), the pressure difference between pressure and suction surface is constantly higher than the other conditions throughout the entire streamwise locations.

The surface pressure distribution recorded at 95% spanwise location (close to the shroud) shows slightly different behaviour compared to mid-span location. It can be seen in Figure 5.30(c) that the pressure difference is higher for all pt1, pt2 and pt4 at 95% span then at 50% span. Furthermore, their magnitudes are now almost similar where no clear differences can be seen except close to the leading and trailing edge of the blade. Therefore, this behaviour indicates that at this particular spanwise location, neither steady nor pulsating flow have the advantage in terms of the power generation capability. The other feature indicated in Figure 5.30 is that the surface pressure difference for pt3 drops very quickly downstream 70% streamwise location.

The analysis from this section yielded several interesting observations. The observation indicated that the most distorted pressure distribution on the blade surface occur at the suction surface close to the rotor hub. Multiple flow separations that lower the power generation capability at this region have been detected. Moreover, the turbine is capable of generating more power at mid-span of the rotor blade during pulsating flow operation as compared to its steady counterpart. However, close to the shroud, neither steady nor pulsating flow operations have the advantage in terms of the power generation capability.
Chapter 5. Pulsating flow turbine operation

5.9 Conclusion

This chapter has presented the analyses conducted for turbine operations under pulsating flow conditions. The CFD results have been validated with experimental data, thus ensuring validity of the model. The assessment of phase shifting methodology has been conducted where the summation of bulk flow and sonic velocity method has been chosen for further analysis.

The effect that pulsating flow have on the incidence angle distribution has been discussed. It has been found that the incidence angle is not at its optimum value during peak pressure ratio condition during a pulse. The comparison between steady and pulsating flow angle distribution has revealed that the turbine operation during pressure decrement instance is closer to its equivalent steady state condition than during pressure increment instance. Furthermore, a reverse flow has been observed at the proximity of the volute tongue during pressure increment instance. Such behaviour does not exist during pressure decrement instance.

The review on the quasi-steady assumption that are widely used in the engine matching process has indicated that there it is necessary to adjust the steady turbine map to accommodate for pulsating flow conditions. Furthermore, analysis using the Λ parameter has yielded that the highest mass imbalance due to pulsating flow occur at pulsating frequency of 60 - 70 Hz.

The final part of the analysis look at the pressure distribution on the surface of the rotor blade. It has been found that the flow suffers multiple separations close to the rotor hub regardless of its operating conditions. Further observation also indicated that the power generation of the blade at mid-span is greater during pulsating flow conditions than steady flow conditions.

In light of the analyses reported in this chapter and the previous chapter, it is possible to make necessary adjustments upstream of the rotor inlet in order to assist the inlet distribution flow angle. In this research, an effort has been made to significantly reduce the number of the vanes to reduce flow blockage at high pressure.
ratio, and assist the flow turning at low pressure ratio. The development of this new concept is explained in details in the following chapter.
Chapter 6

Efficiency improvement through aggressive nozzle vane reduction

6.1 Introduction and background theory

This chapter discusses the efficiency gain recorded by the turbine under pulsating flow conditions with a modified inlet stator. The discussions include the design motivations, modification procedures as well as both steady and pulsating turbine test results.

The analysis conducted in Chapter 4 and 5 revealed not only certain flow features within the turbocharger turbine stage, but also a potential design improvement. In the full vaned configurations, as the pressure ratio increased, it became more difficult for the vanes to turn such high momentum flow in a short distance to its desired flow angle. In addition, during high pressure ratio operations, large periodic deviation of flow angle at the rotor inlet are recorded. However, the gain achieved by the use of vanes cannot be underestimated since it is responsible to even out the flow angle at the volute exit which was originally designed under free vortex assumption. Therefore, without the nozzle vanes, the distribution of the flow angle is expected to be uneven especially during pulsating flow conditions.
There has been disagreement between researchers about the effectiveness of having vanes for better flow guidance towards rotor leading edge. In the application specific to radial or mixed flow turbine, where the flow at vane inlet is already swirled by the volute scroll, the function of vanes is simply to remove any circumferential non-uniformity in the flow. In some cases the vanes are also required to assist the flow turning into the rotor especially at off-design conditions. A classical literature was provided by Baines and Lavy (1990) through their experimental data where a vaned stator indicated higher peak efficiency at a certain operating condition. Other than that the efficiency dropped significantly as compared to its vaneless counterpart. Conversely, recent investigation by Spence et al. (2007) has indicated that the vaneless volute was capable of achieving higher efficiency at all operating conditions compared to the vaned volute.

For these reason, a modification that involve significant reduction of nozzle vanes is attempted. The new design used the aggressively reduced number of vanes (only 30% of the original vanes) arranged symmetrically around the volute circumference. The original motivation behind this modification is to avoid high level blockage during high pressure ratio operations, improve flow angle distribution at volute outlet, as well as mildly turn the flow closer to its optimum incidence at low pressure operations.

Simpson et al. (2013) investigated the influence of nozzle solidity towards the turbine performance. Simpson et al. (2013) found that there exist optimum number of nozzle in order to achieve maximum efficiency. However, for the current research, the numbers of vane blades are much lower than previously investigated by Simpson et al. (2013). It is believed that only a small number of vanes are needed to guide the flow particularly at high pressure ratio while at the same time not introducing extra blockage. Not much guidance is needed at low pressure ratio since the flow velocity is relatively low and the volute alone is sufficient to guide the flow in its intended direction. Furthermore this modification does not involve an active mechanism thus maintaining its simplicity. Experimental testing for both steady and pulsating conditions are conducted to investigate the performance achieved by this new concept. The testing facility and its associated measurement methodology are
described in Chapter 2.

6.2 Development process

The turbocharger turbine used in current work is specifically built for research purpose (Rajoo, 2007). Therefore, modification of its components is made simple (see Figure 6.1). As such, if one of the components is modified and used with the other existing components, the effect of such modification can be clearly evaluated and compared to its original configuration.

![Figure 6.1: The removable vanes and pivoting mechanisms](image)

In the interest of abbreviation, from this point onward this new concept is called reduced vanes concept (RV). In order to evaluate the performance of RV, a few modifications have been made to the original full-vaned (V) arrangement. Since the original vanes are individually pivoted into each hole at the shroud wall of the volute, it is possible to remove and replace the vanes with a specially designed plug in order to seal the unused pivot holes. It is even possible to remove all the vanes to create a vaneless arrangement (VL) as a baseline model for the comparison. The geometry of the plug is shown in Figure 6.2. The plug is designed in such a way that
the leakage flow is kept to the minimum as the pressure ratio increases (see Figure 6.3).

Figure 6.2: Geometry of the plug

Figure 6.3: Assembly of the plug onto the shroud wall

The testing program conducted to test the workability of RV volute is done in conjunction with two other vane arrangements, namely vaneless and full vaned (after this will be referred to as ‘vaned’) configurations. This step is necessary in order to evaluate the efficiency differences of the new volute concept with the more conventional arrangements. The assembly of the plug to form V, VL and RV volute configurations are shown in Figure 6.4, Figure 6.5 and Figure 6.6 respectively.
Figure 6.4: The arrangement of a vaneless volute

Figure 6.5: The arrangement of a full vaned volute
Chapter 6. Efficiency improvement through aggressive nozzle vane reduction

6.3 Review of experimental data

6.3.1 Steady state experimental data

This section presents the steady state performance parameters of all the tested turbine configurations. One of the key to ensure fair comparison is to match the swallowing capacity for all turbine arrangements. This is achieved by regulating the vane angle to match the mass flow parameter recorded by the vaneless volute. Testing is done at the turbine speed of 50% and 80%.

Figure 6.6 shows the comparison of the steady state turbine performance parameter between vaneless, vaned and RV turbine at 50% design speed. It can be seen in Figure 6.7(a) that the swallowing capacity has been matched between the three turbine arrangements, thus allowing direct comparison of turbine efficiency. By comparing the efficiency between the different arrangements (Figure 6.7(b)), it is obvious that the RV volute shows significant improvement for most of the operating condition range. However, at large velocity ratio (> 0.88) the efficiency of RV drops below that of vaned arrangement. Nevertheless, these operating points correspond to low pressure ratio and therefore should not have negative influence during pulsating
Figure 6.7: Steady state performance map of (a) Mass flow parameter and (b) Efficiency for V, VL and RV volute at 50% speed
flow operations.

Figure 6.7(b) also shows that the peak efficiency point tends to shift towards higher velocity ratio values as the number of vanes increases. At 50% speed, the peak efficiency point moves from velocity ratio of about 0.55 to 0.68 as the vanes are added (Figure 6.7(b)). This behaviour could be understood with a closer look at changes that occurred in the velocity triangle plot (see Figure 6.8).

Figure 6.8: The effect of nozzle vanes to the rotor inlet velocity triangle at similar incidence angle

With the nozzle row (green lines) available, the flow area of the turbine inlet is reduced. Therefore, to maintain similar mass flow rate with the vaneless (black lines) volute, the meridional velocity has to increase in order to meet the mass flow continuity requirement (see Equation 6.1).

\[
\rho C_{mv} A_v = \rho C_{mvl} A_{vl}
\]  

(6.1)

In this equation subscript \(v\) indicated vaned volute and \(vl\) indicated vaneless volute. Assuming peak efficiency point occurs at constant optimum incidence angle\(^1\) i.e. relative flow angle (\(\beta_v = \beta_{vl}\)), a higher absolute flow angle is therefore required (\(\alpha_v > \alpha_{vl}\)). At similar turbine speed (\(U_v = U_{vl}\)), this results in the reduction of tangential absolute velocity component (\(C_{\theta v} < C_{\theta vl}\)) which subsequently results in

\(^1\)optimum incidence is in the range of -20\(^\circ\) to -30\(^\circ\) according to Japikse and Baines (1994) where this condition corresponds to peak efficiency point in the turbine map
Chapter 6. Efficiency improvement through aggressive nozzle vane reduction

lower available power into the turbine wheel in accordance with Euler turbomachinery equation (see Equation [6.2]).

\[ W_x = U_2C_{\theta 2} - U_3C_{\theta 3} \]  

(6.2)

where point 2 and 3 correspond to the rotor inlet and outlet respectively. By assuming zero swirl at turbine exit \((U_3C_{\theta 3} = 0)\), Equation [6.2] is reduced to,

\[ W_x = U_2C_{\theta 2} \]  

(6.3)

Subsequently, this produced lower isentropic velocity, \(C_{is}\) and therefore higher velocity ratio\(^{1}\). This behaviour is also valid for RV where the peak efficiency point occurs at velocity ratio between vaned and vaneless arrangement. Figure [6.7(b)] also indicated that with regards to vaned and vaneless arrangement, vaned volute has the advantage at higher velocity ratio \( (> 0.65)\). At low velocity ratio (or high pressure ratio) vaneless volute shown better efficiency as compared to that of vaned volute, but still lower than RV arrangement. This indicated contradictory observations than Spence et al. (2007) in terms of the actual benefit of vaneless volute in radial turbine over its vaned counterpart. These contradictions are possibly due to different turbine types (radial and mixed flow turbine) as well as different method used to match the turbine swallowing capacity with their vaneless counterpart. In addition, the current research used eddy current dynamometer where the true aero-dynamic efficiency is measured by the reaction of the gimbal bearing (chapter 2). In contrast Spence et al. (2007) used compressors wheels as the loading device in order to obtain the turbine efficiency.

The comparisons of steady state performance are slightly different at 80% turbine speed. Figure [6.9(a)] shows that the swallowing capacity for every arrangement has been matched as close as possible to each other. The efficiency profile for this particular speed is plotted in Figure [6.9(b)]. In this plot, it can be seen that throughout all the experimental operating condition, the efficiency of vaneless volute is the lowest,

\[ VR = \frac{V}{e_{sr}} \]

6.3. Review of experimental data
Figure 6.9: Steady state performance map of (a) Mass flow parameter and (b) Efficiency for Vaneless, Vaned and RV volute at 80% speed
followed by vaned volute. The RV arrangement recorded highest efficiency amongst
the three. This observation highlights the benefit of RV at both speed lines specifi-
cally at 80% operating speed where the turbine performance exceeds the vaned and
vaneless turbine at all operation points.

Following the positive efficiency improvements of RV over vaned and vaneless config-
urations under steady state conditions, the testing program was extended to include
the pulsating flow conditions.

6.3.2 Pulsating flow experimental data

The measurement and instrumentation techniques are explained earlier in Chapter
2. The primary aim of this testing programme is to investigate whether the ad-
vantages of RV that are seen during steady state conditions remain during actual
engine operation (pulsating flow conditions). The pulsating flow testing were con-
ducted at three specified loading conditions for two averaged turbine speeds of 30000
rpm and 48000 rpm (50% and 80% design speed respectively). The turbine loading
(power output) were predetermined by adjusting the gap distance between the dy-
namometer permanent magnet and the stator plates (see Table 6.1). Therefore in the
calculations involving averaged parameters later in this chapter, the averaged power
output is used as a reference. Particular attention is also given to the mass flow rate
measurement so that accurate comparisons of the mass flow hysteresis loop between
the different volute arrangements could be made. For RV and vaned arrangement,
the vane angle is kept to the same angle as in steady state conditions. Since the
mass flow parameter for all arrangements are kept to the same level during steady
flow condition, any deviation of recorded instantaneous mass flow parameter during
unsteady testing could be attributed directly to the introduction of unsteadiness of
the flow to a different stator configurations.
Table 6.1: Stator gap distance for pulsating flow testing

<table>
<thead>
<tr>
<th>Stator Gap(mm)</th>
<th>30000 rpm</th>
<th>48000 rpm</th>
</tr>
</thead>
<tbody>
<tr>
<td>Low Load (LL)</td>
<td>7</td>
<td>6.5</td>
</tr>
<tr>
<td>Medium Load (ML)</td>
<td>5</td>
<td>4.5</td>
</tr>
<tr>
<td>High Load (HL)</td>
<td>3.75</td>
<td>3.5</td>
</tr>
</tbody>
</table>

6.3.2.1 Cycle-averaged efficiency evaluation

A simple method to represent the turbocharger turbine efficiency under pulsating flow condition is to calculate its cycle-averaged efficiency. The cycle-averaged efficiency is calculated using Equation 6.4. This method ensures any dependencies on phase shifting by the parameters are eliminated.

\[
\eta_{cyl,avg} = \frac{\sum_{i=1}^{n_{cycle}} \eta(t)_i \cdot W_{isent}(t)_i}{\sum_{i=1}^{n_{cycle}} W_{isent}(t)_i} = \frac{\sum_{i=1}^{n_{cycle}} W_{act(t)_i}}{\sum_{i=1}^{n_{cycle}} W_{isent(t)_i}} \tag{6.4}
\]

As previously explained, the testing are conducted in such a way that the turbine loading is fixed by adjusting the gap distance between the stator and magnet rotor (see Chapter 2 for details on experimental facility). Therefore, at a particular gap setting, the torque obtained is similar if the turbine is rotating at similar speed. This serve as the basis of the cycle-averaged efficiency comparison at different loading condition by plotting it against cycle-averaged actual power. This parameter is calculated using Equation 6.5.

\[
W_{act,avg} = \frac{\sum_{i=1}^{n_{cycle}} W_{act(t)_i}}{n_{cycle}} \tag{6.5}
\]

Figure 6.10 shows the plot of cycle-averaged frequency against the averaged power
Figure 6.10: Plot of cycle-averaged efficiency for all volute configurations against its actual power (turbine loading) at 30000 rpm and 40 Hz pulsating frequency output for all volute configurations at 50% design speed and 40 Hz pulsating frequency. It is obvious from Figure 6.10 that throughout the operation range, RV volute indicated highest cycle-averaged efficiency followed by vaneless volute arrangement. The vaned arrangement shows the lowest cycle-averaged efficiency value which is 6 efficiency points lower than RV at the highest turbine loading. It can also be seen that the new concept of RV produced a constantly higher cycle-averaged efficiency at any loading conditions. However, the benefit of RV volute seems to be more apparent as the turbine loading increases. At the lowest loading, only 1 efficiency point improvement is recorded against full vaned and vaneless volutes. Meanwhile, the vaned volute seems to have the advantage over the vaneless volute only at low loading. As the loading increases, vaneless volute works more efficiently than the vaned volute. Furthermore, the behaviour of the efficiency trend of vaneless and full vaned volute seems to be similar to their steady state behaviour at similar operating speed (see Figure 6.7(b)) where the vaneless volute only shows the efficiency superiority at high loading condition (low velocity ratio). As for the RV volute, the cycle-averaged efficiency shows different trend than its steady state counterpart in terms of the comparison between other arrangements in such a way
that it maintains its higher efficiency in all loading conditions (at 30000 rpm turbine speed).

Figure 6.11: Plot of cycle-averaged efficiency for all volute configurations against its actual power (turbine loading) at 48000 rpm and 40 Hz pulsating frequency

Figure 6.11 shows the plot of cycle-averaged efficiency at 48000 rpm and 40 Hz flow frequency for all volute arrangements. It is a similar type of plot as Figure 6.10 but at different turbine speed. At this speed, it can be seen that at this speed the differences of cycle-averaged efficiency between each volute arrangement are relatively small as compared to 30000 rpm. Although the differences have been reduced, RV volute maintains its superior efficiency over the entire range of loading as compared to the other two arrangements. At low loading condition, the improvement of the cycle-averaged efficiency for RV is recorded to be 4 efficiency points. However, this improvement reduced as the load is increased. At highest loading condition, only 1 efficiency point improvement is recorded. Meanwhile, the behaviour of cycle-averaged efficiency for vaneless and vaned volutes is close to each other except at the maximum loading. While generating the cycle-averaged actual power output of 9 kW, the cycle-averaged efficiency for full vaned volute exceeds that of vaneless volute by 3 efficiency points. The cycle-averaged data for other operating frequencies at 50% and 80% turbine speeds are detailed in Table 6.2 and Table 6.3.
Table 6.2: Cycle-averaged efficiency magnitude for all test conditions at 30000 rpm turbine speed

<table>
<thead>
<tr>
<th></th>
<th>VL</th>
<th>V</th>
<th>RV</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>LL</td>
<td>ML</td>
<td>HL</td>
</tr>
<tr>
<td>20 Hz</td>
<td>0.43</td>
<td>0.60</td>
<td>0.64</td>
</tr>
<tr>
<td>40 Hz</td>
<td>0.45</td>
<td>0.59</td>
<td>0.64</td>
</tr>
<tr>
<td>60 Hz</td>
<td>0.51</td>
<td>0.61</td>
<td>0.62</td>
</tr>
</tbody>
</table>

Table 6.3: Cycle-averaged efficiency magnitude for all test conditions at 48000 rpm turbine speed

<table>
<thead>
<tr>
<th></th>
<th>VL</th>
<th>V</th>
<th>RV</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>LL</td>
<td>ML</td>
<td>HL</td>
</tr>
<tr>
<td>20 Hz</td>
<td>0.34</td>
<td>0.54</td>
<td>0.62</td>
</tr>
<tr>
<td>40 Hz</td>
<td>0.36</td>
<td>0.57</td>
<td>0.68</td>
</tr>
<tr>
<td>60 Hz</td>
<td>0.38</td>
<td>0.57</td>
<td>0.70</td>
</tr>
</tbody>
</table>

Table 6.4: Efficiency difference between RV volute with vaneless and full vaned volute arrangements (%) at 30000 rpm turbine speed

<table>
<thead>
<tr>
<th></th>
<th>$\eta_{RV} - \eta_{VL}$</th>
<th>$\eta_{RV} - \eta_{V}$</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>LL</td>
<td>ML</td>
</tr>
<tr>
<td>20 Hz</td>
<td>4.59</td>
<td>2.21</td>
</tr>
<tr>
<td>40 Hz</td>
<td>1.21</td>
<td>3.08</td>
</tr>
<tr>
<td>60 Hz</td>
<td>-4.37</td>
<td>2.80</td>
</tr>
</tbody>
</table>
Table 6.4 shows the cycle-averaged efficiency differences between the RV and the other two volute arrangements for all testing frequencies and loadings at 50% turbine speed. Positive values indicate efficiency improvement while negative values indicate efficiency deficit. It can be seen that RV concept indicates higher efficiency than vaneless and vaned in almost all test conditions except for two conditions, both of which are at low loading and high frequency (60 Hz). Maximum efficiency improvement that is achieved is 7.71 and 8.19 efficiency points for vaneless and vaned volute respectively. The magnitude of efficiency improvement of RV at this speed seems to be substantial as it is traditionally not easy to improve the efficiency of the turbine during pulsating flow environments. Nevertheless, there are also conditions where the application of RV is not favourable. At low loading and 60 Hz pulsating frequency, the RV volute registered the efficiency deficit of -4.37 and -3.02 efficiency points compared to vaneless and full vaned volutes respectively.

Table 6.5: Efficiency difference between RV volute with vaneless and full vaned volute arrangements (%) at 48000 rpm turbine speed

<table>
<thead>
<tr>
<th></th>
<th>$\eta_{RV} - \eta_{VL}$</th>
<th>$\eta_{RV} - \eta_{V}$</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>LL  ML  HL</td>
<td>LL  ML  HL</td>
</tr>
<tr>
<td>20 Hz</td>
<td>6.80  5.25  0.03</td>
<td>1.92  2.02  -1.17</td>
</tr>
<tr>
<td>40 Hz</td>
<td>4.35  1.08  0.74</td>
<td>1.48  1.17  -1.79</td>
</tr>
<tr>
<td>60 Hz</td>
<td>4.89  0.03  -3.22</td>
<td>1.66  -2.31  -1.75</td>
</tr>
</tbody>
</table>

Table 6.5 shows the efficiency difference between RV volute and vaneless as well as vaned volutes at 48000 rpm turbine speed. It can be seen that as compared to the efficiency difference at 30000 rpm (Table 6.4), RV concept still shows higher efficiency at most of the testing conditions. The maximum improvement of efficiency is recorded against vaneless volute at low loading and 20 Hz flow frequency with the magnitude of 6.80 efficiency points. According to Table 6.5, it can also be deduced that as the speed increases, the efficiency improvements of RV tend to favour low loading conditions. This observation is evidence as at 48000 rpm, RV volute suffer efficiency penalty against full vaned volute at high loading conditions for all pulsating frequencies. Furthermore it also suffers efficiency penalty at high loading condition...
at 60 Hz flow frequency if compared against the vaneless volute. Nevertheless, the comparison at high loading for 48000 rpm is somehow difficult to assess as the turbine produced significantly different actual power output at high turbine loading even though the stator-to-rotor gap is similar. Perhaps the cycle-averaged efficiency is best represented by a 3 dimensional plot with actual power and flow frequency as its x-axis and y-axis (see Figure 6.12 and 6.13).

![Figure 6.12: Map of efficiency difference between RV and vaneless/full vaned volutes at all testing conditions for 50% design speed](image)

![Figure 6.13: Map of efficiency difference between RV and vaneless/full vaned volutes at all testing conditions for 80% design speed](image)

In general, the RV concept has met the initial objective to achieve more power
Chapter 6. Efficiency improvement through aggressive nozzle vane reduction

extraction than vaned and vaneless volute arrangement. This is shown in both steady and unsteady turbine performance efficiency map where at most tested conditions, the RV volute is able to achieve either similar or greater efficiency than other tested arrangements.

It has been indicated earlier that the vane angle settings during pulsating flow testing are kept the same as it is during steady flow testing\textsuperscript{1}. However, during pulsating flow operations, it has been found that the instantaneous swallowing differs between each volute arrangement. The instantaneous swallowing capacity behaviour for RV, as well as its differences to the other arrangements are detailed in the next section.

6.3.2.2 Swallowing capacity characteristics

Figure 6.14: Plot of mass flow parameter against pressure ratio at 50% speed and 20 Hz flow frequency for different volute configurations

Figure 6.14 shows the instantaneous mass flow parameter plot against the instantaneous pressure ratio at 20 Hz pulsating frequency for all three volute configurations. Also plotted as black lines in the same axis in Figure 6.14 is the equivalent steady

\textsuperscript{1} at a constant speed, the steady state swallowing capacity of V, VL and RV is matched to each other by means of vane angle
state turbine swallowing capacity map at similar speed of 30000 rpm. It can be seen that the hysteresis loops were formed for the instantaneous plot of mass flow parameter for all volute arrangements, and all of them are clockwise in direction. For the vaneless volute, as the pressure start to increase, the swallowing capacity also increases and closely matches its equivalent steady flow swallowing capacity up until pressure ratio of 1.26. As the pressure climbs beyond the particular pressure ratio, the swallowing capacity seems to deviate from its steady flow equivalent to higher magnitude. This behaviour is the indication that the air is filling the volute at higher rate than the steady state condition, also known as ‘filling’. At this point forward towards higher pressure ratio until the maximum point, the relationship between the instantaneous mass flow parameter is almost linear with the instantaneous pressure ratio. The development of instantaneous pressure ratio and mass flow rate later reach the peak at similar instance (1.55 pressure ratio), resulting in sharp reversal hysteresis at the top right of the figure. Subsequently, there is a rapid decrement of swallowing capacity as the pressure ratio begins to decrease, albeit its magnitude is still higher than the equivalent steady flow mass flow parameter curve. Slower rate of pressure ratio change as compared to the change of mass flow parameter suggests that the effect of the reflected compression wave. As the pressure continues to decline, the instantaneous mass flow parameter matched its steady state curve at 1.33 pressure ratio only to decrease even lower as pressure goes down in a pulse. The behaviour of the instantaneous mass flow parameter to fall below the steady state condition is the indication of mass emptying the volume of the volute and that the rate of mass transfer into the volute is less due to previously available mass still leaving the volute.

As indicated in Figure 6.14 the swallowing capacity hysteresis loop for the volute with full vanes configuration exhibits a noticeably different shape than that of vaneless configuration. During the pressure build up at the beginning of the pulse, the instantaneous mass flow parameter deviates away to higher magnitude than the steady state conditions, and therefore filling the volute. The relationship between the swallowing capacity and pressure ratio continues to be almost linear until both parameters reach their peak at similar instance. This behaviour, also observed in vaneless configuration, suggests that the reflection of the compression wave (with
any surface downstream) from the preceding pulse has yet to reach the measurement plane as the current pressure peaks. Close to the peak, as the pressure ratio begins to decrease, the instantaneous swallowing capacity seems to be decreasing quicker for just a short moment (much less than the case of vaneless volute) before maintaining almost linearly decreasing relationship with the pressure ratio. This behaviour implies that full vane arrangement exhibit little reflection from the preceding compression wave at this particular operating point of 30000 rpm and 20 Hz pulsating frequency. As the pressure ratio decreases to its minimum value, the mass flow parameter shows lower magnitude than the equivalent steady state condition, suggesting so called ‘emptying’ effect (also observed in vaneless volute). It is clearly seen from Figure 6.14 that the hysteresis loop of the swallowing capacity for full vaned volute is relatively narrower than the other two arrangements. From the observation of this two turbine arrangements so far, it can be deduced that the introduction of pulsating flow could have the effect of increasing mass flow parameter beyond its steady state limit.

The qualitative description for the behaviour of the swallowing capacity hysteresis for the RV arrangement is somewhat a combination of vaneless and full vaned volutes. The filling process is still visible at lower end of pressure ratio as the pressure starts to climb. As the pressure increases even more, there is an almost linear relationship between the two parameter plotted in Figure 6.14 similar behaviour as explained for the other volutes before. The swallowing capacity for RV also peaks at similar instance as the pressure ratio, therefore a sharp edge is created. However, as the pressure ratio reduces, the mass flow parameter drops substantially, forming a loop that resembles the one observed for the vaneless volute. This behaviour serves an as an indication of the existence of reflected compression wave from preceding pulse that reached the measurement plane after the current pulse peaks, therefore slowing the pressure decrement rate. However, unlike the vaneless volute that maintain the loop until the pressure ratio reaches minimum value, this loop seems to merge at pressure ratio of 1.35, therefore resembles the full vaned hysteresis behaviour. However, this merging part is only temporary before the instantaneous mass flow parameter drops lower than its steady state equivalent, thus indicating emptying process of the volute.

6.3. Review of experimental data
Figure 6.15: Plot of instantaneous mass flow parameter against pressure ratio at 50% speed and 40 Hz frequency for different volute configurations

Figure 6.15 shows the instantaneous swallowing capacity of all three volute arrangements at 30000 rpm and 40 Hz pulsating frequency as well as their steady state equivalent conditions which is plotted as dotted line. As the pulsating frequency is increased from 20 Hz to 40 Hz, distinct changes in the trend of the development of swallowing capacity hysteresis can be seen (see Figure 6.14 and Figure 6.15). For the vaneless volute, as the pressure begins to rise the swallowing capacity increment seem to be slower than the steady state conditions until the pressure ratio of 1.25 where it matches each other. Beyond the particular instance, the mass flow rate increase at higher rate than that of steady state conditions, indicating the start of filling process. The instantaneous swallowing capacity reaches its maximum point at 1.45 pressure ratio on which the pressure ratio is still climbing before reaching its peak at 1.48 pressure ratio. The increasing measured pressure even though the mass flow already decreasing originated from the reflection of the compression wave of succeeding pulse with the turbine components downstream. Such behaviour does not occur at 20 Hz pulsating frequency (see Figure 6.14) thus resulting in different shape of the instantaneous swallowing capacity shape. Subsequently, the pressure and mass flow parameter decrease at almost similar rate as their climbing rate, therefore maintaining the width of the loop. As the pressure ratio fall below 1.25
to its minimum value, the swallowing capacity seems to have retained its value and merge with steady state line.

Meanwhile, for the full vaned volute, the mass flow parameter and pressure ratio relationship climbs almost linearly but reach higher value of mass flow parameter than the vaneless at its peak. For this configuration the hysteresis also indicated that the swallowing capacity reaches its peak value than the pressure ratio. The possible reason for this behaviour is explained before with vaneless volute at similar operating condition. It can also be seen that the pressure ratio retain its magnitude close to the peak thus resulting a ‘round’ edge of the hysteresis at top right of the figure. The subsequent decrement of pressure ratio is accompanied with decrement of mass flow parameter at similar magnitude with vaneless volute until about 1.27 pressure ratio. At the pressure ratio lower than this point, the mass flow parameter climbed to the magnitude close to its steady state equivalent and before reduced again to its minimum value, creating secondary loop at low pressure ratio region. Albeit the loop is visible and measurable, the fact that it occurs at relatively such low pressure ratio indicates its secondary importance in the overall performance of the turbine.

Perhaps, more interesting observation of the swallowing capacity loop is that of the RV at 40 Hz pulsating frequency where it shows mixing of feature between both other volutes. This observation was indeed also recorded at 20 Hz frequency. The hysteresis loop of RV follows closely to the vaneless hysteresis during instances where the pressure is climbing. Similar to the other two arrangements, the mass flow parameter reached its peak before the pressure ratio. However, as soon as the pressure drops, the hysteresis trend of RV immediately follows the full vaned hysteresis curve right until its minimum pressure ratio. Therefore, from the observation at these two frequencies (20 Hz and 40 Hz), it is clear that the hysteresis loop of instantaneous swallowing capacity form RV follows that of vaneless as the pressure ratio climbs and subsequently follows that of full vaned volute as the pressure ratio drops.

Figure 6.16 shows the plot of instantaneous mass flow parameter against pressure ratio for all three volute configurations at 30000 rpm and pulsating frequency of 60
Figure 6.16: Plot of instantaneous mass flow parameter against pressure ratio at 50% speed and 60 Hz frequency for different volute configurations.

Hz. A clear distinct feature that is recorded for the particular frequency is that the swallowing capacity loops for vaneless and RV volutes has formed intersecting loops which have not been seen for lower frequency cases. Similar feature also does not appear for full vaned configuration at 60 Hz. For the vaneless volute, the increment of instantaneous pressure ratio is accompanied by the increment of swallowing capacity, albeit below its steady state equivalence until up to 1.25 pressure ratio. As the pressure ratio increases beyond that, the mass flow parameter indicated filling process where the rate of mass flow increment is higher than the quasi steady conditions. The mass flow parameter and pressure ratio then reaches their peak magnitude at similar instance. The subsequent feature shows that the formation of hysteresis loop happens in counter-clockwise direction, a uniquely shaped only observed at 60 Hz flow frequency. This indicates that the rate of decreasing pressure ratio is higher than the rate of decreasing swallowing capacity, therefore suggesting that the reflected compression wave from the preceding pulse has reached the current pulse before its peak. This situation of counter-clockwise hysteresis however, does not persist until minimum pressure ratio. The loop seems to be intersecting at pressure ratio of 1.29 thus forming a normal clockwise-type hysteresis again. The magnitude falls below steady state equivalence which indicates the emptying effect and
finally formed a secondary loop at minimum pressure ratio. However as discussed before, the existence of the secondary loop is not central to the turbine performance since it occurs only during low pressure ratio instance where less isentropic power is available.

Despite the new trend of hysteresis recorded for vaneless volute, the shape of the loop for the full vaned volute at similar frequency exhibit a rather different behaviour. This can be seen right from the beginning of the pulse where the increment of instantaneous swallowing capacity follow the steady state increment closely until 1.21 pressure ratio. Filling process take part beyond this point where the increment of mass flow parameter is almost linear with the increment rate of pressure ratio. These two parameters reach their peaks at similar instant. As the pressure decrease, a hysteresis with increasing width is formed. This behaviour is not recorded in other frequency cases where the loop is either maintaining its width (40 Hz) or collapsing (20 Hz) as the pressure ratio decrease. As the pressure ratio reaches its minimum magnitude, the swallowing capacity tends to follow its steady state equivalent conditions, therefore causing the hysteresis to collapse.

The swallowing capacity hysteresis for RV volute indicated some interesting feature on which it follows the loop of vaneless volute during pressure increment period and the loop of full vaned volute during pressure decrement period. Similar behaviour is also seen in previous cases with 20 Hz and 40 Hz flow frequencies. Since the hysteresis loop for vaneless and full vaned volute at 60 Hz shows significantly different behaviour (see Figure 6.16) as discussed above, the discussion for RV volute is only focused on the main difference. In RV, the instantaneous mass flow parameter increases following closely the curve of vaneless volute until the maximum pressure ratio. The mass flow parameter for RV, however, does not follow the counter-clockwise loop of vaneless volute instead decrease at almost similar rate as it increases. This seems to be the combined effect of mass flow decrement rate of vaneless and full vaned volute. As the pressure ratio reaches its minimum value, the mass flow parameter tends to move towards its steady state conditions, just as seen in full vaned volute arrangement.
Chapter 6. Efficiency improvement through aggressive nozzle vane reduction

The observations from Figure 6.14 to Figure 6.16 indicated that the geometrical properties of the volute influenced the progression of the instantaneous mass flow parameter hysteresis loop even though similar rotor wheel are used, and the swallowing capacity is matched during steady state condition. Therefore, the application of rotor of different size and geometrical properties are expected to have even more influence towards the hysteresis trend.

Figure 6.17: Plot of instantaneous swallowing capacity for RV volute arrangement at 30000 rpm and 40 Hz flow frequency under different loading conditions

Figure 6.17 shows the development of hysteresis loop in RV volute at 30000 rpm and 40 Hz flow frequency as the loading is increased. The definitions of Low, Medium and High loading and their properties are described in Table 6.1. In general Figure 6.17 indicated that the range of instantaneous mass flow parameter and pressure ratio increases as the loading increase. This is expected as the amplitude of the pulse will increase as more loading is applied to the turbine. However, it can also be seen that the range of both parameters is not the only change that occur as the loading increases. The increment in loading also affects the shape of the loop which subsequently means that the reflection and superposition of the travelling pressure waves are also affected. At low loading condition, the increment of instantaneous swallowing capacity clearly indicates filling where the loop is above the steady state.
condition. The pressure and mass flow parameter achieve peak at similar instance where the loop subsequently as pressure ratio decrease and mass flow parameter reduces below the steady-state line. It is also worth noting that the minimum pressure ratio recorded during unsteady operations is significantly lower than the minimum pressure ratio during steady state conditions, therefore indicating that at certain instance the turbine has to operate at near-to-ambient pressure ratio. Furthermore for this particular condition, the hysteresis loop shows no sign of significant reflection of the compression wave from the succeeding pulse.

In terms of the swallowing capacity hysteresis shape, narrower hysteresis is produced as the load increases from low to medium load (see Figure 6.17). As the pressure ratio increases, the swallowing capacity increases below its steady state equivalence until pressure ratio of 1.22. Subsequently, filling process takes effect. At this particular loading condition, it can be seen that the pressure ratio reaches its maximum magnitude before the swallowing capacity. This feature indicated that the compression wave from the succeeding pulse has reflected and reached the measurement plane just as the current pulse are about to peak. This effects the development of the hysteresis loop in such a way that a counter-clockwise loop is formed. However, it is not long after mass flow parameter has reached its peak that the loop intersects each other and regains the normal clockwise formation until the pressure ratio reaches its minimum again.

The shape of hysteresis loop changes again as the loading is increased from medium to high. At this condition, the loop is larger, indicating significant filling and emptying. Perhaps the most obvious distinct feature from the other loading occurs at the higher end of pressure ratio. This is similar plot as explained earlier in Figure 6.15. Therefore, only a brief discussion is provided here. The increment of loading from medium to high has altered the point where the compression wave reflected back to the measurement plane. As the mass flow reaches its peak earlier than the pressure ratio, it can be deduced that the reflected compression wave only reach the current pulse short after its peak, resulting in delayed peaking of instantaneous pressure ratio. The remaining development of the swallowing capacity hysteresis loop has been described in Figure 6.15.
Figure 6.18: Plot of instantaneous swallowing capacity for RV volute arrangement at 48000 rpm and 40 Hz flow frequency under different loading conditions

It has now been established that the alteration in turbine loading not only effect the range of pressure pulse amplitude, but also effect its travelling patterns. However, similar observation ceases to exist at 80% turbine speed (48000 rpm). Figure 6.18 shows the plot of instantaneous swallowing capacity against pressure ratio for RV volute at 80% turbine speed. Also plotted in the same axis is the equivalent steady state mass flow parameter at similar speed. It is clear from this figure that the features that exist in the loop at one loading also exist at another loading. Two most apparent similarities for all the loops are that the instantaneous mass flow parameter reaches its maximum value before the pressure ratio. Moreover, as the pressure ratio returns to its minimum value, there is an indication that the mass flow parameter tends to be close to its steady state equivalence. This observation shows that the development of mass flow parameter hysteresis loop does not only depend on the flow frequency and loading, but also on the turbine speed.

Analysis on the instantaneous swallowing capacity for different volute arrangements revealed the differences that is not seen during steady state conditions. The analysis also revealed that the instantaneous swallowing capacity for RV arrangement resembles the vaneless volute during pressure increment period and full vaned volute
during pressure decrement period. In addition, the shape of these hysteresis loops depends on the loading, turbine speed, turbine geometry as well as flow frequency.

As discussed earlier in chapter 5, the mass flow changes within the domain relative to the total mass flow at the boundary could be represented as \( \Lambda \). Evaluation of this parameter for different turbine arrangement is the main focus of the next section.

### 6.3.2.3 Lambda parameter

\( \Lambda \) parameter is effectively the ratio of time averaged of the changes of mass flow in the turbine stage against time averaged mass flow of the through flow. It is used to evaluate the level of mass imbalance in the turbine stage where the value of 1 indicated highly unsteady flow. In this chapter, the extensive testing procedures to evaluate the unsteady performance comparison between RV, vaneless and vaned volute at multiple operating conditions have provided an opportunity to further see the behaviour of this parameter. The full summary of calculated Strouhal number (St), Pi Parameter (\( \Pi \)) and \( \Lambda \) parameter is detailed in Table 6.6 and 6.7.
### Table 6.6: Values of Strouhal, Π and Λ for 50% turbine speed

<table>
<thead>
<tr>
<th>Speed</th>
<th>Load</th>
<th>f</th>
<th>St</th>
<th>Π</th>
<th>A</th>
<th>St</th>
<th>Π</th>
<th>A</th>
<th>St</th>
<th>Π</th>
<th>A</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td></td>
<td>Vaneless</td>
<td></td>
<td></td>
<td></td>
<td>Vaned</td>
<td></td>
<td></td>
<td></td>
<td>RV</td>
</tr>
<tr>
<td>50%</td>
<td>Low</td>
<td>10</td>
<td>0.1517</td>
<td>0.1503</td>
<td>0.0228</td>
<td>0.1469</td>
<td>0.1862</td>
<td>0.0274</td>
<td>0.1470</td>
<td>0.1603</td>
<td>0.0236</td>
</tr>
<tr>
<td></td>
<td></td>
<td>20</td>
<td>0.3007</td>
<td>0.1717</td>
<td>0.0516</td>
<td>0.2895</td>
<td>0.1951</td>
<td>0.0565</td>
<td>0.3098</td>
<td>0.1795</td>
<td>0.0556</td>
</tr>
<tr>
<td></td>
<td></td>
<td>40</td>
<td>0.6010</td>
<td>0.1413</td>
<td>0.0849</td>
<td>0.5881</td>
<td>0.1824</td>
<td>0.1073</td>
<td>0.5858</td>
<td>0.1650</td>
<td>0.0967</td>
</tr>
<tr>
<td></td>
<td></td>
<td>60</td>
<td>0.9791</td>
<td>0.1435</td>
<td>0.1405</td>
<td>0.8983</td>
<td>0.1597</td>
<td>0.1434</td>
<td>0.8474</td>
<td>0.1578</td>
<td>0.1337</td>
</tr>
<tr>
<td>50%</td>
<td>Med</td>
<td>20</td>
<td>0.2343</td>
<td>0.2621</td>
<td>0.0614</td>
<td>0.2356</td>
<td>0.2918</td>
<td>0.0688</td>
<td>0.2283</td>
<td>0.2879</td>
<td>0.0657</td>
</tr>
<tr>
<td></td>
<td></td>
<td>40</td>
<td>0.4702</td>
<td>0.1970</td>
<td>0.0926</td>
<td>0.4567</td>
<td>0.2420</td>
<td>0.1105</td>
<td>0.4670</td>
<td>0.2240</td>
<td>0.1046</td>
</tr>
<tr>
<td></td>
<td></td>
<td>60</td>
<td>0.6863</td>
<td>0.1777</td>
<td>0.1220</td>
<td>0.6673</td>
<td>0.2024</td>
<td>0.1350</td>
<td>0.6911</td>
<td>0.1940</td>
<td>0.1341</td>
</tr>
<tr>
<td>50%</td>
<td>High</td>
<td>20</td>
<td>0.1908</td>
<td>0.3690</td>
<td>0.0704</td>
<td>0.1843</td>
<td>0.3980</td>
<td>0.0733</td>
<td>0.1854</td>
<td>0.4151</td>
<td>0.0770</td>
</tr>
<tr>
<td></td>
<td></td>
<td>40</td>
<td>0.3701</td>
<td>0.3133</td>
<td>0.1160</td>
<td>0.3561</td>
<td>0.3309</td>
<td>0.1178</td>
<td>0.3699</td>
<td>0.3244</td>
<td>0.1200</td>
</tr>
<tr>
<td></td>
<td></td>
<td>60</td>
<td>0.5459</td>
<td>0.2081</td>
<td>0.1136</td>
<td>0.5185</td>
<td>0.2316</td>
<td>0.1201</td>
<td>0.5486</td>
<td>0.2317</td>
<td>0.1271</td>
</tr>
</tbody>
</table>
Table 6.7: Values of Strouhal, II and Λ for 80% turbine speed

<table>
<thead>
<tr>
<th>Speed</th>
<th>Load</th>
<th>f</th>
<th>St</th>
<th>Π</th>
<th>Λ</th>
<th>St</th>
<th>Π</th>
<th>Λ</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td></td>
<td>Vaneless</td>
<td></td>
<td></td>
<td>Vaned</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>Speed</td>
<td>Load</td>
<td>f</td>
<td>St</td>
<td>Π</td>
<td>Λ</td>
</tr>
<tr>
<td>20</td>
<td>Low</td>
<td>0.2209</td>
<td>0.3176</td>
<td>0.0702</td>
<td>0.2225</td>
<td>0.3435</td>
<td>0.0764</td>
<td>0.2246</td>
</tr>
<tr>
<td>40</td>
<td>Low</td>
<td>0.4415</td>
<td>0.2396</td>
<td>0.1058</td>
<td>0.4303</td>
<td>0.2775</td>
<td>0.1194</td>
<td>0.4486</td>
</tr>
<tr>
<td>60</td>
<td>Low</td>
<td>0.6696</td>
<td>0.1706</td>
<td>0.1142</td>
<td>0.6375</td>
<td>0.1920</td>
<td>0.1224</td>
<td>0.6587</td>
</tr>
<tr>
<td>20</td>
<td>Med</td>
<td>0.1825</td>
<td>0.3964</td>
<td>0.0724</td>
<td>0.1818</td>
<td>0.4448</td>
<td>0.0809</td>
<td>0.1873</td>
</tr>
<tr>
<td>40</td>
<td>Med</td>
<td>0.3639</td>
<td>0.3209</td>
<td>0.1168</td>
<td>0.3543</td>
<td>0.3452</td>
<td>0.1223</td>
<td>0.3657</td>
</tr>
<tr>
<td>60</td>
<td>Med</td>
<td>0.5419</td>
<td>0.2124</td>
<td>0.1151</td>
<td>0.5330</td>
<td>0.2218</td>
<td>0.1182</td>
<td>0.5296</td>
</tr>
<tr>
<td>20</td>
<td>High</td>
<td>0.1583</td>
<td>0.5057</td>
<td>0.0801</td>
<td>0.1586</td>
<td>0.5795</td>
<td>0.0919</td>
<td>0.1578</td>
</tr>
<tr>
<td>40</td>
<td>High</td>
<td>0.3168</td>
<td>0.4545</td>
<td>0.1440</td>
<td>0.3133</td>
<td>0.4226</td>
<td>0.1324</td>
<td>0.3197</td>
</tr>
<tr>
<td>60</td>
<td>High</td>
<td>0.4722</td>
<td>0.2367</td>
<td>0.1118</td>
<td>0.4557</td>
<td>0.2556</td>
<td>0.1165</td>
<td>0.4635</td>
</tr>
</tbody>
</table>
In Table 6.6 and 6.7, it can be seen that the highest magnitude of the \( \Lambda \) parameter is recorded at 40 Hz frequency during high loading condition for the vaneless volute. At this condition, the mass imbalance within the turbocharger turbine stage is recorded to be 14.4%. The following discussion details the behaviour of \( \Lambda \) parameter for different speed and loading conditions.

![Figure 6.19: Plot of Lambda parameter for different volute arrangements at 30000 rpm turbine speed under low loading condition](image)

Figure 6.19 shows the plot of \( \Lambda \) parameter for low loading conditions for all the volute configurations. In general, at this particular operating condition, the value of \( \Lambda \) parameter increases with the frequency. The comparison between all three volute arrangements indicated that full vaned configuration has the highest value of \( \Lambda \) parameter throughout the operating frequencies. Vaneless volute recorded lowest \( \Lambda \) parameter value at the frequency of 10 Hz, 20 Hz and 40 Hz but has higher \( \Lambda \) parameter than the RV volute at the highest frequency.

Similar plot of \( \Lambda \) parameter for medium loading operations is shown in Figure 6.20. The trend of the development of \( \Lambda \) parameter seems to be similar between low and medium load operation. However, in Figure 6.20, the \( \Lambda \) parameter value for vaneless volute remains the smallest among the configurations for the entire flow frequencies.
6.3. Review of experimental data

Figure 6.20: Plot of Lambda parameter for different volute arrangements at 30000 rpm turbine speed under medium loading condition

Figure 6.21: Plot of Lambda parameter for different volute arrangements at 30000 rpm turbine speed under high loading condition
Figure 6.21 shows the plot of Λ parameter at high loading condition. There is a significant change in the development of Λ parameter at this condition as compared to the other loading at similar turbine speed (Figure 6.19 and Figure 6.20). At high loading, the increment in Λ parameter value is not similar between configurations as the frequency increases. The order of Λ parameter magnitude has also changed where RV is now indicated highest value throughout the frequencies. This is followed by full vaned and vaneless volutes. For RV, it is clear that the increment in Λ parameter value reduced significantly at frequency higher than 40 Hz. Despite that, the magnitude of Λ parameter for RV at 60 Hz is still higher than its magnitude at 40 Hz. Meanwhile, for the full vaned volute, the Λ parameter that initially increased at similar rate as the other volutes from 20 Hz to 40 Hz indicated similar value of Λ parameter between 40 Hz and 60 Hz flow frequency. This feature has been recorded earlier in Chapter 5 where the Λ parameter increased at low frequency and then stabilized frequency higher than 40 Hz, albeit the magnitude is a lot higher due to large pressure amplitude. Perhaps, the most interesting observation in Figure 6.21 is that of vaneless volute. The Λ parameter recorded for this volute peaks at 40 Hz and then decreases at higher frequency. This observation indicated that after a certain pressure ratio (turbine loading), the high mass imbalance in the turbocharger turbine tend to occur at lower frequency.

Figure 6.22: Plot of Lambda parameter for different volute arrangements at 48000 rpm turbine speed under high loading condition

6.3. Review of experimental data
At 48000 rpm, the value of Λ almost always peaks at 40 Hz flow frequency for each volute configuration as tabulated Table 6.7. Only two exceptions occur at low loading condition where RV and vaneless volute recorded their highest Λ at 60 Hz frequency. A particularly interesting plot for the particular speed is recorded at high loading condition. At this point the Λ value for vaneless volute increased substantially between 20 Hz to 40 Hz thus exceeding the value recorded with RV and full vaned configurations. Subsequently, the Λ value for the particular volute reduced to the same level as RV and full vaned volutes at 60 Hz flow frequency.

Analysis of the Λ parameter magnitude in this section has indicated different trend of development at different turbine speed and loading. Due to this differences, full characterization of the turbine ‘unsteadiness’ is made difficult. Despite that, the data presented is sufficient in order to conclude that the mass imbalance in the turbine stage is not necessarily higher at high frequency, regardless of its geometry (V, VL or RV).

6.4 Conclusion

This chapter presents the efficiency improvement achieved through aggressive reduction of the nozzle vanes. It has been shown that the new concept is capable of performing better than its full vaned and vaneless counterpart for both steady and unsteady conditions, with the exception of a few operating conditions. A thorough analysis on the instantaneous swallowing capacity behaviour has been conducted. The analysis has shown that for different geometrical configurations, the similar swallowing capacity achieved between each other during steady flow operations does not guarantee similar instantaneous swallowing capacity during pulsating flow operation.
Chapter 7

Closure

This thesis presents the development of a three-dimensional computational model to assist understanding of the flow field behaviour within the turbocharger turbine stage under steady and pulsating flow conditions. Experimental work has also been conducted to provide crucial validation information for the numerical model. Furthermore, a novel concept, achieved by aggressive reduction of nozzle vanes, turbine has been developed and tested. The comparison with vaneless and full vaned turbine yielded improved efficiency at similar level of swallowing capacity. This chapter will include the main conclusions from the research, as well as recommendations for future work.

7.1 Conclusions

• Successful development of an accurate full-stage three dimensional CFD model for the turbocharger turbine stage operating under pulsating flow conditions.

The Reynolds-Averaged Navier Stokes (RANS) equation is solved using the commercial software *Ansys CFX 14.0*. The closure issue due to the Reynolds Stress terms in the RANS is modelled using the k-epsilon turbulence model. The computational do-
main consists of 4 major components which are the inlet duct, the volute, the vanes and the rotor wheel. Precise geometrical properties that correspond to actual testing conditions are modelled and meshed using various commercial software including Solidworks, ICEM CFD and Turbogrid. Particular attention is given towards modelling the vanes geometry as the manufacturing inaccuracy has resulted in slight changes of the tip gap. It was found that this area has to be modelled precisely in accordance with actual operating condition and not the original dimensions in order for the validation data to be usable. The components of the domain are assembled in Ansys CFX Pre where the boundary conditions obtained from experimental data are applied.

The experimental data are obtained using the cold-flow turbocharger test facility available at Imperial College London. This facility is built as a representative of an actual hot turbine operating conditions in a vehicle. This is made possible by employing similarity approach. The facility is capable of generating pulsating flow with the desired pressure profile that represents opening and closing of the actual exhaust valve in a vehicle. The employment of eddy current dynamometer allows direct measurement of the actual power using the turbine torque, therefore reducing the dependency on accurate temperature measurement. Furthermore, the flexibility to adjust the spacing between the magnet rotor and stator within the dynamometer allows larger turbine operating range to be tested as compared to the conventional compressor as the loading device, resulting in wider turbine map. Details of experimental instrumentation particularly for unsteady testing are also described. Finally, the uncertainty analysis is explained which shows that the experimental uncertainty becomes large as the pressure ratio is reduced due to sensitivity of the load cell.

For the steady state simulations, the predictions of mass flow parameter matched with experimental data obtained on the test-rig with the average deviation of 2%. The steady state turbine efficiencies are also well predicted with similar average deviation value of 2%. All the predicted points fall below the experimental uncertainty limits. Unlike the validation exercises for steady state condition that used the derived parameters such as turbine efficiency and mass flow parameter, the validations for unsteady calculations are done using the primary parameters, namely
static pressure and torque. For both of the parameters, the magnitude of peaks and troughs of the instantaneous plots agree well with experimental data, albeit slight shifting in the phase angle of the predicted results are recorded close to the peak for both parameters.

- In-depth understanding of the turbine behaviour under steady and pulsating flow conditions and its relationship with internal flow field structure.

1. Steady state conditions

For the steady state conditions, the analysis at the maximum efficiency operation indicated that the existence of vanes has reduced the circumferential variation of the flow angle that exit the volute (from $\pm 10^0$ to $\pm 20^0$). The averaged incidence flow angle is $-10^0$ which is close to the optimum magnitude. Although the circumferential variation of flow angle at the rotor inlet is limited by the existence of the vanes, there still exist large variations of velocity across the inlet span where the effect of tip leakage flow is evidence. As the inlet pressure increased or decreased, the deviation of incidence angle from the optimum value is not symmetrical. This in turn results in the asymmetrical plot of total losses ($1 - \eta$) against incidence angle. Results also indicated that the circumferential fluctuations of flow angle increases as the inlet pressure increases.

2. Pulsating flow conditions - phase shifting method

For the pulsating flow conditions, the locations on which the isentropic and actual power are evaluated are different and therefore, phase shifting is necessary. The assessment of phase shifting procedure is conducted where the shifting technique using the summation of bulk flow and sonic velocity has been proven to be better than simply matching the peak of isentropic and actual power.

3. Pulsating flow conditions - flow field analysis

7.1. Conclusions
It is clear from the analysis that the incidence angle is highly positive as the pressure ratio peaks. This prompted the instantaneous efficiency of the turbine to be very low at instances where most of the power is available. As the speed increases, the maximum instantaneous incidence angle is reduced albeit still indicated positive values. From this particular analysis, it can be concluded that the possibility of increasing the cycle-averaged efficiency of the turbine of a fixed geometry lies in the design that is capable of operating efficiently at instances where most of the energy are available.

Analysis at 1.6 PR indicated that the circumferential variation of the relative flow angle is multiplied by more than seven times during unsteady conditions as compared to its steady state counterpart. The comparison between the pressure increment and decrement period for all flow frequencies suggested that the circumferential distribution of the flow angles are more chaotic during pressure increment. The range of flow angle distribution during pressure increment period is recorded to be more than twice as compared to similar parameter during pressure decrement period.

4. *Pulsating flow conditions - turbine ‘unsteadiness’*

The calculations of Lambda parameter are also presented in the analysis. The parameter represents ‘unsteadiness’ of the turbine by including the effect of both pulsating flow frequency and its amplitude. It has been found that the development of lambda parameter for both 50% and 80% turbine speed is almost similar in term of its trend as the frequency increases, albeit a small shifting to the higher value as the speed reduces from 48000 rpm to 30000 rpm. Therefore in general, it can be concluded that the turbine behaviour is more ‘unsteady’ at lower speed as compared to higher speed. In addition, potential prediction errors by simply assuming that the turbine wheel under unsteady operating environment behaves as a quasi-steady device are also discussed.
• Successful development of the reduced vanes stator concept that results in efficiency increment of the current turbine under steady and pulsating flow conditions.

As the name implies, this concept is achieved by aggressively reducing the number of the nozzle vanes to only a third of its original number. In accessing the influence of the new stator arrangement towards the turbocharger performance, comparison is made with both full vaned (15 blades) and vaneless volute arrangements. The steady state swallowing capacity map for all the volute configurations are matched by regulating the vane angle, therefore the difference in performance is only influenced by the stator geometry. Experimental testing are conducted at both steady and pulsating flow operating conditions.

For steady state turbine operations, the application of the new concept yields clear improvement over both full vaned and vaneless arrangements. At 50% speed, the maximum efficiency achieved by the reduced vanes concept (RV) is 73% which is 4 efficiency points higher than the maximum efficiency of the other arrangements. However, slight efficiency deficit (2 efficiency point) is recorded compared to the full vaned configuration at velocity ratio higher than 0.89. At 80% speed the efficiency improvement is not so clear but the RV is constantly more efficient throughout all the operating points than the other configurations. Also noted in the experimental efficiency map is the shifting in peak efficiency point towards higher velocity ratio value as the number of vanes increases.

In unsteady operating conditions, the main parameter used to evaluate the turbine performance is the cycle-averaged efficiency. For operation under pulsating flow environment, the new RV concept has shown positive cycle-averaged efficiency improvement that is as high as 8 efficiency point as compared to the other arrangements. The performance improvement is more obvious at 50% turbine speed than 80% speed. There are a few conditions on which the RV has recorded loss in efficiency particularly at 80% speed and high loading conditions. The maximum efficiency deficit that is recorded is 4.3 efficiency points. As for the swallowing capacity hysteresis loop of RV, it is found that the RV loop follows the trend of
the loop recorded on the vaneless volute during pressure increment period. On the other hand, during pressure decrement period, the loops follow that of full vaned arrangement.

### 7.2 Future work

#### 7.2.1 Understanding of the turbine behaviour under pulsating flow condition

In order to fully understand the turbine behaviour under pulsating flow conditions, much work needed to be done. It would be beneficial to have the performance data for the turbine under more flow frequencies and turbine loadings (pressure amplitude). This would allow accurate mapping of the evolution of the turbine behaviour and its dependency on both frequencies and amplitude of the pressure profile.

It would also be interesting to investigate the effect of a single pulse towards the turbine performance. This will help in isolating the effect of a purely pulsating flow by eliminating the superposition of pulses that is usually seen at high flow frequency. However, the main difficulty to produce a single pulse is to build a sufficiently fast valve to provide equivalent pulse timescale as the tested conditions.

Although CFD could provide accurate predictions and also enable detailed flow analysis, there are needs for much faster model to assist turbine designers and engineers to perform turbocharger-engine matching procedure by taking advantage of pulsating flow operations. It is recommended that the available CFD data be used as an integral part to assist the prediction capability of the low order models.
7.2.2 Improvement of the new nozzle vanes arrangement

The testing done for the RV volute in this research has shown promising results. However, more investigations are needed to optimize its geometrical configuration. Asymmetrical nozzle arrangement could be tested in order to obtain optimized configuration. Moreover, the testing program in the current research employed a fixed vane angle in order to match the mass flow parameter to the other turbine configurations. As it has been proven to be able to achieve higher efficiency than its counterparts, it is essential to investigate the RV performance at different vane angle position. Different turbine geometry could also be coupled to the RV in order to prove the global benefit of this new concept.
References


References


References  

221


Appendices
Appendix A

V-Cone calibration document

Figure A.1: V-Cone calibration document
Appendix B

Bezier curve definition

In constructing the blade profiles (Chapter 3), Bezier polynomials is used to generate smooth curves following a few defining points of Rotor A (see Table B.1 by Palfreyman (2004)). The resulting cylindrical points from this polynomials will define the leading edge, trailing edge, hub, and shroud of the rotor. Subsequently these coordinates are transformed into Cartesian coordinates that are later used in Ansys Turbogrid software.
Table B.1: Rotor A defining characteristics

<table>
<thead>
<tr>
<th>Geometric feature</th>
<th>Notation</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Radius of hub at leading edge [mm]</td>
<td>$R_1$</td>
<td>36.008</td>
</tr>
<tr>
<td>Radius of hub at trailing edge [mm]</td>
<td>$R_2$</td>
<td>13.535</td>
</tr>
<tr>
<td>Blade height at leading edge [mm]</td>
<td>$B_1$</td>
<td>17.990</td>
</tr>
<tr>
<td>Blade height at trailing edge [mm]</td>
<td>$B_2$</td>
<td>25.790</td>
</tr>
<tr>
<td>Axial length of turbine [mm]</td>
<td>$L$</td>
<td>40.000</td>
</tr>
<tr>
<td>Cone angle [°]</td>
<td>$\delta$</td>
<td>50.000</td>
</tr>
<tr>
<td>Theta-coordinate of leading edge root [°]</td>
<td>$\theta_0$</td>
<td>-10.768</td>
</tr>
<tr>
<td>Theta-coordinate of leading edge control point [°]</td>
<td>$\theta_1$</td>
<td>-4.963</td>
</tr>
<tr>
<td>Z-coordinate of leading edge control</td>
<td>$Z_1$</td>
<td>6.545</td>
</tr>
<tr>
<td>Trailing edge blade angle [°]</td>
<td>$\beta$</td>
<td>-67.740</td>
</tr>
<tr>
<td>1\textsuperscript{st} Bezier parameter (camber-line)</td>
<td>$P_c$</td>
<td>0.500</td>
</tr>
<tr>
<td>2\textsuperscript{nd} Bezier parameter (camber-line)</td>
<td>$Q_c$</td>
<td>0.500</td>
</tr>
<tr>
<td>Theta-coordinate of trailing edge [°]</td>
<td>$\theta_4$</td>
<td>-25.000</td>
</tr>
<tr>
<td>1\textsuperscript{st} Bezier parameter (hub)</td>
<td>$P_h$</td>
<td>0.500</td>
</tr>
<tr>
<td>2\textsuperscript{nd} Bezier parameter (hub)</td>
<td>$Q_h$</td>
<td>0.500</td>
</tr>
<tr>
<td>1\textsuperscript{st} Bezier parameter (shroud)</td>
<td>$P_s$</td>
<td>0.500</td>
</tr>
<tr>
<td>2\textsuperscript{nd} Bezier parameter (shroud)</td>
<td>$Q_s$</td>
<td>0.500</td>
</tr>
<tr>
<td>Blade number</td>
<td>$Z$</td>
<td>12</td>
</tr>
<tr>
<td>(Tip-gap)/(blade height + tip gap)</td>
<td>$T_g$</td>
<td>0.030</td>
</tr>
</tbody>
</table>
Appendix B. Bezier curve definition

Leading edge

\[
\begin{align*}
\theta &= (1 - u)^2 tle0 + 2u(1 - u) tle1 + u^2 tle2 \\
z &= (1 - u)^2 zle0 + 2u(1 - u) zle1 + u^2 zle2
\end{align*}
\]  \hspace{1cm} (B.1)

where

\[
\begin{align*}
tle0 &= \theta_0 \\
tle1 &= \theta_1 \\
tle2 &= 0.0 \\
zle0 &= 0.0 \\
zle1 &= Z_1 \\
zle2 &= B_1 \cdot \sin(\delta)
\end{align*}
\]  \hspace{1cm} (B.2)

\[
\begin{align*}
SOle &= \frac{tle1 - tle0}{zle1 - zle0} \\
S2le &= \frac{tle2 - tle1}{zle2 - zle1} \\
C2le &= \frac{1}{2} \frac{(zle2)(tle0 - tle1) - (tle0)(zle1)}{(zle2 - zle1)^3}
\end{align*}
\]
Appendix B. Bezier curve definition

Camber line

\[
\theta = (1 - u)^4 tc10 + 4u(1 - u)^3 tc1 + 6u^2(1 - u)^2 tc2 + 4u^3(1 - u)tc3 + u^4 tc4
\]
\[
z = (1 - u)^4 zc10 + 4u(1 - u)^3 zc1 + 6u^2(1 - u)^2 zc2 + 4u^3(1 - u)zc3 + u^4 zc4
\]

(B.3)

where

\[
S0cl = S2le
\]
\[
C0cl = C2le
\]
\[
S4cl = \tan(\beta) \left( \frac{\pi}{180} \right)
\]
\[
tcl0 = 0.0
\]
\[
tcl4 = \theta_4
\]
\[
zc10 = B_1 \sin(\delta)
\]
\[
zcl4 = L
\]
\[
zc = \frac{tcl4 + (S0cl)(zc10) - (S4cl)(zcl4)}{S0cl - S4cl}
\]

(B.4)
\[
tc = S0cl(zc - zc10)
\]
\[
J = \left( \frac{4}{3} \right) \frac{(C0cl)(p_{.c})^2(zc10 - zc)^3}{(zc10)(tc - tcl4) + (zc)(tcl4) - (zcl4)(tc)}
\]
\[
tcl1 = (P_c)(tc)
\]
\[
tcl2 = tc + (J)(tcl4 - tc)
\]
\[
tcl3 = tcl2 + (Q_c)(tcl4 - tcl2)
\]
\[
zcl1 = zc10 + (P_c)(zc - zc10)
\]
\[
zcl2 = zc + (J)(zc4 - zc)
\]
\[
zcl3 = zcl2 + (Q_c)(zcl4 - zcl2)
\]
Appendix B. Bezier curve definition

Hub curve

\[ r = (1-u)^4 rh0 + 4u(1-u)^3 rh1 + 6u^2(1-u)^2 rh2 + 4u^3(1-u) rh3 + u^4 rh4 \]
\[ z = (1-u)^4 zh0 + 4u(1-u)^3 zh1 + 6u^2(1-u)^2 zh2 + 4u^3(1-u) zh3 + u^4 zh4 \]  \hspace{1cm} (B.5)

where

\[ S0h = -\tan(\delta) \]
\[ rh0 = R_1 \]
\[ rh1 = R_1 + (P_h)(R_2 - R_1) \]
\[ rh2 = rh3 = rh4 = R_2 \]
\[ zh0 = 0.0 \]
\[ zh1 = (P_h) \left( \frac{R_2 - R_1}{S0h} \right) \]  \hspace{1cm} (B.6)
\[ zh2 = \frac{R_2 - R_1}{S0h} \]
\[ zh3 = \left( \frac{R_2 - R_1}{S0h} \right) + (Q_h) \left( L - \left( \frac{R_2 - R_1}{S0h} \right) \right) \]
\[ zh4 = L \]
Appendix B. Bezier curve definition

Shroud curve

\[ r = (1 - u)^4rs_0 + 4u(1 - u)^3rs_1 + 6u^2(1 - u)^2rs_2 + 4u^3(1 - u)rs_3 + u^4rs_4 \] \[ z = (1 - u)^4zs_0 + 4u(1 - u)^3zs_1 + 6u^2(1 - u)^2zs_2 + 4u^3(1 - u)zs_3 + u^4zs_4 \] (B.7)

where

\[ S_0s = S_0h = -\tan(\delta) \]

\[ rs_0 = R_1 + \frac{(B_2)(\cos(\delta))}{1 - T_g} \]

\[ rs_2 = rs_3 = rs_4 = R_2 + \frac{b_2}{1 - T_g} \]

\[ rs_1 = rs_0 + (P_s)(rs_4 - rs_0) \] (B.8)

\[ zs_0 = \frac{(B_1)\sin(\delta)}{1 - T_g} \]

\[ zs_2 = zs_0 + \frac{rs_4 - rs_0}{S_0s} \]

\[ zs_1 = zs_0 + (P_s)(zs_2 - zs_0) \]

\[ zs_4 = L \]

\[ zs_3 = zs_2 + (Q_s)(zs_4 - zs_2) \]
Appendix C

Additional test results

Figure C.1: Instantaneous mass flow parameter at 30000 rpm, 20 Hz, high loading condition
Appendix C. Additional test results

Figure C.2: Instantaneous mass flow parameter at 30000 rpm, 40 Hz, high loading condition

Figure C.3: Instantaneous mass flow parameter at 30000 rpm, 60 Hz, high loading condition
Figure C.4: Instantaneous mass flow parameter at 30000 rpm, 20 Hz, medium loading condition

Figure C.5: Instantaneous mass flow parameter at 30000 rpm, 40 Hz, medium loading condition
Figure C.6: Instantaneous mass flow parameter at 30000 rpm, 60 Hz, medium loading condition

Figure C.7: Instantaneous mass flow parameter at 30000 rpm, 20 Hz, low loading condition
Appendix C. Additional test results

Figure C.8: Instantaneous mass flow parameter at 30000 rpm, 40 Hz, low loading condition

Figure C.9: Instantaneous mass flow parameter at 30000 rpm, 60 Hz, low loading condition
Appendix C. Additional test results

Figure C.10: Instantaneous mass flow parameter at 48000 rpm, 20 Hz, high loading condition

Figure C.11: Instantaneous mass flow parameter at 48000 rpm, 40 Hz, high loading condition
Appendix C. Additional test results

Figure C.12: Instantaneous mass flow parameter at 48000 rpm, 60 Hz, high loading condition

Figure C.13: Instantaneous mass flow parameter at 48000 rpm, 20 Hz, medium loading condition
Appendix C. Additional test results

Figure C.14: Instantaneous mass flow parameter at 48000 rpm, 40 Hz, medium loading condition

Figure C.15: Instantaneous mass flow parameter at 48000 rpm, 60 Hz, medium loading condition
Figure C.16: Instantaneous mass flow parameter at 48000 rpm, 20 Hz, low loading condition

Figure C.17: Instantaneous mass flow parameter at 48000 rpm, 40 Hz, low loading condition
Figure C.18: Instantaneous mass flow parameter at 48000 rpm, 60 Hz, low loading condition