Enhancement of wind turbine performance by installation above a bluff body

Goh, Ernest Seach Chyr

2014

Goh, E. S. C. (2014). Enhancement of wind turbine performance by installation above a bluff body. Doctoral thesis, Nanyang Technological University, Singapore.

<https://hdl.handle.net/10356/62052>

<https://doi.org/10.32657/10356/62052>

**ENHANCEMENT OF
WIND TURBINE PERFORMANCE BY
INSTALLATION ABOVE A BLUFF BODY**

GOH SEACH CHYR (ERNEST)

School of Mechanical & Aerospace Engineering

A thesis submitted to the Nanyang Technological University
in partial fulfilment of the requirements for the degree of
Doctor of Philosophy

2014

ACKNOWLEDGEMENTS

I would like to express sincere thanks to my supervisor Assistant Professor Jörg Uwe Schlüter, who has patiently guided me throughout the three years of this research. I especially thank him for giving me the freedom to design and pursue the research as I thought fit.

I would also like to express heartfelt thanks to the Directors of the Energy Research Institute, Professors Chan Siew Hwa and Subodh Mhaisalkar, for giving me this golden opportunity, in spite of my age, to pursue a PhD course while working as a research associate. Without the financial stability that the research position gave me, it would not have been possible for me to pursue studies while supporting my family at the same time.

My appreciation also goes out to all those who have helped me at some time or other during the course of my research: Ji Xiaona, my senior for helping me with background literature and ANSYS software; technicians Seow Tzer Fook, Chia Yak Khoong and Elson Ng Moo Kheo for their assistance with equipment and prototypes; project officers Vincent Chai Wee Sern and Sethu Raman Boopathy, intern Chidambaresan Krishnaswami; final year project students Muhammed Al-Muzakkir and Edwin Ong Han Ze for their assistance during various phases of experiments and office mates Arne Reinecke, Wilbur Tan Hong Huat and Jerry Xiang Junting for helping to proof-read this thesis.

I also thank my wife and children for their kind understanding whenever I had to sacrifice time with them due to research commitments.

Last but not least, I would like to thank the Maritime and Port Authority of Singapore for generously funding this research.

CONTENTS

ACKNOWLEDGEMENTS	i
LIST OF FIGURES	vi
LIST OF TABLES	xiii
NOMENCLATURE AND LIST OF SYMBOLS	xv
ABSTRACT	xvi
Chapter 1. INTRODUCTION	1
1.1 Background	1
1.2 Objectives	1
1.3 Hypothesis	2
1.4 Thesis Outline	3
Chapter 2. LITERATURE REVIEW	5
2.1 Wind: a Viable Source of Renewable Energy	5
2.2 Urban Wind Energy	5
2.3 Drag- & Lift-Based Cross-Flow Wind Turbines	5
2.4 Airflow Modifications for Cross-Flow Turbines.....	12
2.5 Turbulence.....	14
Chapter 3. RESEARCH METHODOLOGY	19
3.1 Overview of Methodology	19
3.2 Experimental Work.....	19
3.2.1 Wind Tunnel Tests	19
3.2.2 Driving Test of Turbine Test Rig.....	21

3.2.3	Choice of Test Period and Time	21
3.3	Computational Fluid Dynamics Simulation	22
3.3.1	Software	22
3.3.2	Validation	24
3.3.3	Independence Studies.....	25
3.3.4	Parametric Studies and Normalisation	26
3.3.5	Establishment of Reference Flow Field.....	27
3.3.6	Comparison with Three Baseline Cases	27
Chapter 4.	EXPERIMENTAL WORK & DISCUSSION OF RESULTS	29
4.1	Overview of Experimental Work	29
4.2	Wind Tunnel Facility	30
4.3	Wind Tunnel Tests with Forward Facing Step.....	30
4.3.1	Design and Method.....	31
4.3.2	Results of Wind Tunnel Step Tests.....	32
4.4	Wind Tunnel Tests with Reduced-Scale Turbine.....	33
4.4.1	Design and Method.....	33
4.4.2	Results of Wind Tunnel Turbine Tests	34
4.5	Driving Tests	35
4.5.1	Design of the Savonius Turbine	35
4.5.2	Design of the Bluff Body.....	37
4.5.3	Power Transmission and Brake	38
4.5.4	Instrumentation	39

4.5.5	Location of Experiment.....	42
4.5.6	Calibration.....	43
4.5.7	Experimental Procedure	45
4.5.8	Tests with Bluff Body Only	46
4.5.9	Driving Test Results and Discussion.....	47
Chapter 5.	CFD SIMULATIONS	53
5.1	Determining Required Apparatus for Wind Tunnel Tests.....	53
5.2	Determining Optimum Turbine Position for Driving Test Rig.....	55
5.3	Validation with Wind Tunnel Test of Wind Turbine.....	61
5.4	Simulation of Driving Test Conditions.....	65
5.4.1	Domain Size and Grid Independence Studies.....	66
5.4.2	Comparison with other turbulence models.....	71
5.4.3	Validation with Driving Test Results	72
5.4.4	Summary of Applicable Simulation Settings.....	79
5.5	Optimisation of a Generic Turbine.....	80
5.5.1	Choice of Step Height and Domain Size.....	80
5.5.2	Validation with Prior Literature	86
5.5.3	Parametric Simulation Results.....	89
5.5.4	Summary of Generalised CFD Methodology	96
Chapter 6.	EMPIRICAL MODEL.....	97
6.1	Development of the Empirical Relationship	97
6.1.1	Overview of the Empirical Relationship.....	97

6.1.2	The Source of Torque Production at Low TSR	98
6.1.3	The Source of Torque Production at High TSR.....	102
6.1.4	Discussion on Optimal turbine Axis position	104
6.2	Normalised Equation of the Line of optimal Installation position.....	110
6.3	Comparison with Baselines	111
Chapter 7.	CONCLUSION & FUTURE WORK.....	114
7.1	Conclusion	114
7.2	Future Work	116
7.2.1	CFD Simulation Work	116
7.2.2	Experimental Work	117
REFERENCES		120
Appendix A		126
Appendix B		128
Appendix C		132

LIST OF FIGURES

Figure 1. Turbine installed above bluff body.	2
Figure 2. Turbine installed above flat surface.	3
Figure 3. Left: horizontal axis cross-flow wind turbine [8]. Right: conventional vertical axis cross-flow wind turbine [9].	6
Figure 4. Solar updraft tower in Manzanares, Spain. Inset: vertical axis axial flow wind turbine inside the base of the tower [10].	6
Figure 5. Lift-based cross-flow turbines: (left to right) World's largest cross-flow wind turbine, the Éole at Cap Chat, Quebec, built according to the original Darrieus design [11], giromill or H-rotor [9], Pison Cycloturbine with phase-dependent and wind-speed-dependent pitch variation [12], helical version of giromill [13].	7
Figure 6. Drag-based cross-flow wind turbines: (clockwise from top left) Three-bladed Savonius used in the present research, helical version of Savonius [14], cup-anemometer-style turbine [15], Ecowing with phase-dependent blade pitch variation [16].	7
Figure 7. Combined drag- and lift-based cross-flow wind turbines: (left to right) Hi-VAWT [17], Lenz turbine [18], Cygnus Power [19].	8
Figure 8. Control volume for Betz's derivation.	8
Figure 9. Schematic of a drag-driven blade.	11
Figure 10. Airflow modification: (clockwise from top left) ducted Honeywell Turbine [29], Donqi diffuser augmented turbine, KR Windpower cross-flow turbine with guide vanes [30], horizontally mounted version of the KR Windpower turbine manufactured by VQ Wind [31].	13
Figure 11. Flowchart of methodology.	20
Figure 12. Average wind monthly wind speeds [38].	22

Figure 13. Diurnal mean wind speed for Pandan Gardens in April 2012 [39].	22
Figure 14. The Workbench Environment.	23
Figure 15. Torque obtained by simulation, clearly showing that for the case with bluff body (BB), settling occurred much later.	25
Figure 16. Parameters that were studied.	26
Figure 17. Baselines. Left: null condition, centre: conventional installation, right: to verify whether null condition is non-optimal.	28
Figure 18. Three parts of experimental work, (left to right) wind tunnel, wind tunnel with reduced-scale turbine, driving test.	29
Figure 19. Closed-loop wind tunnel.	30
Figure 20. Wind tunnel test of step blockage.	31
Figure 21. Comparison of simulation and experimental results of step blockage in wind tunnel, wind speed 8 m/s, step height 0.2 m.	32
Figure 22. Testing of wind turbine above a step blockage in wind tunnel.	33
Figure 23. Wind tunnel test to determine optimum turbine position on the test rig.	34
Figure 24. Exploded view of laminated construction of end plate.	36
Figure 25. Blade placement, showing curvature and overlap.	36
Figure 26. Overall view of the driving test lorry with bluff body and turbine.	37
Figure 27. Braking system.	38
Figure 28. Anemometer position.	39
Figure 29. Photograph of anemometer as installed for driving test.	40
Figure 30. Wheel speed measurement.	41
Figure 31. Digital speedometer with scrolling display in metres per second.	41

Figure 32. Data acquisition by Hantek 1008A PC oscilloscope (centre black box). The DSO-2150 measures brake disc spoke pulses which were not used. The breadboard in the foreground is the brake controller. On the left is the Chatillon force gauge..	42
Figure 33. Driving test site: Lim Chu Kang Road. Inset: satellite view, showing a straight section of 2.4 km.....	43
Figure 34. Calibration of turbine brake load cell.	44
Figure 35. Load cell calibration results.	44
Figure 36. Test vehicle ready for first test.	45
Figure 37. Vehicle with bluff body only.	46
Figure 38. Driving test result, 12 m/s.....	49
Figure 39. Driving test result, 10 m/s.....	50
Figure 40. Driving test result, 8 m/s.....	51
Figure 41. Flow field in wind tunnel with a bluff body installed, showing velocity contour and vector plots. Detailed view of rectangular region is shown in Figure 42.	54
Figure 42. Detailed view of rectangular region in Figure 41 with final mesh after three adaption steps.....	54
Figure 43. Simulation domain and boundary conditions for determining the optimum installation position of a turbine.....	56
Figure 44. Detailed view of the mesh for determining the optimum installation position, element size labelled.	56
Figure 45. A typical torque result from the simulation to determine optimum installation position of turbine on test rig.	57
Figure 46. Simulation result of power of turbine above a bluff body, $y = 1.4$	58
Figure 47 Simulation result of power of turbine above a bluff body, $y = 1.6$	59

Figure 48. Simulation result of power of turbine above a bluff body, $y = 1.8$	59
Figure 49. Velocity flow field of the position with the highest power output: $x = 1.0$ m, $y = 1.6$ m.....	60
Figure 50. Old position of turbine (left) compared to new position (right).	61
Figure 51. CFD simulation of experiments by Saha et al., typical velocity contour plot. ..	63
Figure 52. CFD simulation of experiments by Saha et al., bottom view of domain mesh (partial detailed view).	63
Figure 53. CFD simulation of experiments by Saha et al., typical y^+ plot, wind speed 9.48 m/s, TSR 0.5.	64
Figure 54. CFD simulation of experiments by Saha et al.	64
Figure 55. Typical CFD simulation result showing the velocity contour plot. The blue octahedron indicates the position of the anemometer.	66
Figure 56. Simulation of driving test conditions, variation of wind load and torque with domain size.	67
Figure 57. Simulation of driving test conditions, the three regions that form the simulation domain.	68
Figure 58. Simulation of driving test conditions, overview of mesh (above) and detail of turbine region (below).	69
Figure 59. Simulation of driving test conditions, variation of wind load and torque with grid fineness.	70
Figure 60. Simulation of driving test conditions, typical y^+ , wind speed 12.081 m/s, TSR 0.88306.	70
Figure 61. Simulation of a typical driving test condition using three different turbulence models.....	72
Figure 62. Comparison of CFD and experimental result: 12 m/s.....	73

Figure 63. Comparison of CFD and experimental result: 10 m/s.	74
Figure 64. Comparison of CFD and experimental result: 8 m/s.....	75
Figure 65. Lateral component of prevailing wind.....	76
Figure 66. Sketch of plan view of test rig: no cross wind (top); with cross wind, driving left to right (middle); with cross wind, driving right to left (bottom).....	76
Figure 67. Simulation to support choice of step height: mesh after refinement and boundary conditions.	81
Figure 68. Variation of domain length after the step, from left to right: 4x, 12x, and 20x step height. Note complete inclusion of the separation bubble at 20x.	82
Figure 69. Variation of domain total height, from left to right: 10x, 20x, and 40x step height. Note insignificant difference between 20x and 40x.....	82
Figure 70. Variation of domain length before the step, from left to right: 6x, 9x, and 15x step height. Note disappearance of inflow for 15x case.	82
Figure 71. Contour plots above a forward facing step at 12 m/s free stream wind speed for step height of 10 m (left) 15 m (right).....	83
Figure 72. Simulation to support choice of step height: comparison of velocity profiles for suitable size and 20% larger size.	84
Figure 73. Wind speed profile for step height of 5 m, free stream wind speed of 12 m/s.	84
Figure 74. Wind speed profile for step height of 10 m, free stream wind speed of 12 m/s.	85
Figure 75. Wind speed profile for step height of 15 m, free stream wind speed of 12 m/s.	85
Figure 76. CFD simulation of experiment by Baker, longitudinal velocity component plot. The lines are measurement locations.....	86

Figure 77. CFD simulation of experiment by Baker, showing the mesh after three adaption steps and boundary conditions.	87
Figure 78. Longitudinal velocity from CFD simulation of Baker's work compared with his experimental results.	87
Figure 79. Normalised turbulent kinetic energy from CFD simulation of Baker's work compared with his experimental results.	88
Figure 80. Domain for parametric simulation of generic turbine above a bluff body (forward facing step).....	89
Figure 81. Overview of mesh, generally showing the underside of the domain looking from the outlet end.....	90
Figure 82. Mesh of wake and rotating region.....	90
Figure 83. Typical torque and wind load result. Dotted lines show the average values over one revolution.....	91
Figure 84. Velocity contour plot of the turbine and wake region. View angle: 15° azimuth from turbine axis, 15° elevation.....	91
Figure 85. Pressure contour plot of the turbine and wake region. View angle: 15° azimuth from turbine axis, 15° elevation.....	92
Figure 86. Mesh near turbine surfaces, without inflation (left), with inflation (right).....	93
Figure 87. Wall y^+ without inflation layers (left), with inflation layers (right).	93
Figure 88. Sketch of the turbine positions that were simulated. R is turbine radius (= 1.25 m).	94
Figure 89. c_p vs TSR for a range of turbine positions, for turbine diameter of 2.5 m, bluff body step height of 5 m and free stream wind speed of 12 m/s	95
Figure 90. Line of peak c_p , also the line of optimal installation position.....	96

Figure 91. Line of peak c_p superimposed on wind speed lines, $H = 5$ m, $V_\infty = 12$ m/s, $R = 1.25$ m.	97
Figure 92. Extract from Figure 89, highlighting two peaks in the c_p vs. TSR curve for turbine position $x = 1R$, $y = 2$ m.	99
Figure 93. Torque over one revolution, TSR 0.8.	101
Figure 94. Torque over one revolution, TSR 1.2.	103
Figure 95. Torque plots, pressure & velocity contour plots for turbine position $x = 1R$, y varying from 1.5 m (bottom) to 2.5 m (top).	106
Figure 96. Torque plots, pressure & velocity contour plots for turbine position $x = 2R$, y varying from 2.0 m (bottom) to 3.0 m (top).	107
Figure 97. Torque plots, pressure & velocity contour plots for turbine position $x = 3R$, y varying from 2.5 m (bottom) to 3.5 m (top).	108
Figure 98. Torque plots, pressure & velocity contour plots for turbine position $x = 4R$, y varying from 2.5 m (bottom) to 3.5 m (top).	109
Figure 99. Domain for simulation of a turbine above a flat surface, axis horizontal.	112
Figure 100. For axis horizontal, lower half returning (left) and upper half returning (right).	112
Figure 101. Domain for simulation of a turbine above a flat surface, axis vertical.	112
Figure 102. Baseline results.	113

LIST OF TABLES

Table 1. Typical simulation setup to verify hypothesis.....	2
Table 2. Parameters affecting coefficient of power.	27
Table 3. Uncertainties in experimental measurements.	47
Table 4. Driving test results for 12 m/s vehicle speed. $Re = 8.68 \times 10^5$	49
Table 5. Driving test results for 10 m/s vehicle speed. $Re = 7.24 \times 10^5$	50
Table 6. Driving test results for 8 m/s vehicle speed. $Re = 5.79 \times 10^5$	51
Table 7. ANSYS CFX settings for determining step height in wind tunnel.....	53
Table 8. ANSYS CFX settings for preliminarily obtaining optimum position.....	57
Table 9. Extract of results of wind tunnel tests on three-bladed Savonius tubine by Saha et al.....	62
Table 10. ANSYS CFX settings for simulating the experiment of Saha et al.....	62
Table 11. ANSYS CFX settings for simulating the driving test conditions.	65
Table 12. Comparison of three turbulence models with experimental result.	71
Table 13. CFD simulation result for 12 m/s vehicle speed (also free stream wind speed).	73
Table 14. CFD simulation result for 10 m/s vehicle speed (also free stream wind speed).	74
Table 15. CFD simulation result for 8 m/s vehicle speed (also free stream wind speed).	75
Table 16. Comparison of lateral wind direction of consecutive runs, 12 m/s.....	77
Table 17. Comparison of lateral wind direction of consecutive runs, 10 m/s.....	78
Table 18. Comparison of lateral wind direction of consecutive runs, 8 m/s.....	78
Table 19. Summary of applicable simulation settings.	79

Table 20. ANSYS CFX settings for simulating bluff buildings of various heights, equivalent to a two-dimensional forward facing step.....	80
Table 21. Suitable domain size for simulation of flow over a forward facing step.	83
Table 22. Positions of peak coefficient of power	98

NOMENCLATURE AND LIST OF SYMBOLS

A	area
a	induction factor
C	(with various subscripts) constants in turbulence model equations
c_D	coefficient of drag
c_P	coefficient of power
D	turbine diameter
F_D	drag force
H	bluff body (forward facing step) height
k	turbulent kinetic energy
L	characteristic length
p	(lowercase) pressure
P	(uppercase) power, mean pressure (where stated)
R	turbine radius
S	strain rate tensor of the mean flow
T	thrust
TSR	tip speed ratio, ratio of blade tip speed to free stream wind speed
U	wind speed (mean flow, where applicable)
$\mathbf{U}$	wind velocity vector (mean value)
$\mathbf{u}$	wind velocity vector (total)
$\mathbf{u}'$	wind velocity vector (fluctuating component)
u, v, w	components of wind vector in the three Cartesian directions
V_∞	free stream wind speed
ε	rate of dissipation of turbulent kinetic energy
λ	ratio of blade speed to free stream wind speed, also known as tip speed ratio
μ	dynamic viscosity
μ_t	turbulent viscosity
ν	kinematic viscosity
ρ	density
σ, γ, β	(with various subscripts) constants in turbulence model equations
Φ	general flow property (mean value)
ϕ	general flow property
ϕ'	general flow property (fluctuating component)
ω	specific dissipation rate

ABSTRACT

When a bluff body is exposed to wind resulting in flow over a forward facing step, the flow field above the bluff body consists of a circulating separation bubble. The wind velocity within this bubble varies from being negative (opposite in direction to the free stream wind) near the top surface of the bluff body, to near zero at the centre of the bubble, to high positive in the top part of the bubble before gradually reducing to the free stream wind speed far away from the surface. The high positive wind velocity in this flow field can be exploited to boost wind turbine power output. The present research has discovered that when optimally installed with axis horizontal, a two-bladed Savonius turbine above an infinitely wide step has a coefficient of power (c_P , calculated using the free stream wind speed) which is 2.6 times the c_P of an identical turbine operating in an unbounded airspace.

The optimal turbine installation position is at $0.56H$ above the top surface in the vertical direction and $0.75H$ behind the front edge in the horizontal direction, where H is the step height. Extending from this optimal point is a line of optimal installation position. For a flow over a forward facing step, a line of minimum wind speed, generally passing through the middle of the separation bubble can be extracted from the flow field. The line of optimal installation position is similar in shape to this line of minimum wind speed but offset above it.

The methodology used in the present research consists primarily of Computational Fluid Dynamics (CFD) parametric simulation covering a range of installation positions and turbine tip speed ratio (TSR). The commercial solver ANSYS CFX in the workbench environment was used. Validation of the simulation results were carried out using data from driving tests of a test rig mounted on a lorry driven at a range of speeds. The test rig consists of a Savonius turbine installed above a bluff body. During the design of the test rig, wind tunnel tests and supporting CFD simulations were used to determine the turbine position that is likely to give maximum power output. As a result of the driving tests, practical experience was gained which will be useful for future researchers.

CHAPTER 1. INTRODUCTION

1.1 BACKGROUND

Wind energy has become economically viable, with generation costs almost on par with other technologies [1], both fossil fuel-based and renewable. The number of large scale commercial wind energy projects is growing rapidly. In recent years, interest has spread to urban wind energy projects. Urban wind energy projects have the advantage of generating electricity where it is needed and avoids transmission losses. The use of small wind turbines has become increasingly popular to supplement energy harvesting. Such wind turbines are installed on existing structures, such as the roof of a building.

One of the approaches to harnessing urban wind energy is to ‘transplant’ relatively well established axial flow turbine technology. Another approach is to consider novel turbine designs and integrate them with the buildings on which they are intended to be installed. An example of the latter approach is the WindNok turbine [2]. The present research is similar. However, instead of studying a customised installation such as the WindNok, the present research focuses on exploiting the flow field above a generic cuboidal building to enhance wind turbine performance. Such buildings are common and thus hopefully, the outcome of this research will be applicable to a wider range of situations. The flow structure around the building is not uniform and hence the location of the small wind turbine needs to be done wisely to exploit the non-uniformity.

The author would like to thank the Maritime and Port Authority of Singapore (MPA) for generously funding this research.

1.2 OBJECTIVES

The objectives of this research are:

- 1) To verify the hypothesis that a cross-flow drag-based wind turbine installed above a forward facing step bluff body has a higher power output than one conventionally installed above a flat surface using Computational Fluid Dynamics (CFD) simulation;

- 2) To determine, for one combination of bluff body step height, wind speed and turbine size, the line of optimal installation position that maximises coefficient of power (c_p); and,
- 3) To generalise the methodology so that further work involving other combinations of parameters can be systematically performed.

1.3 HYPOTHESIS

The fundamental starting point of the present research is the hypothesis that operating a wind turbine installed above a bluff body will enhance its performance. This hypothesis has been positively verified by CFD simulation and thus it has been decided to continue with the present research. For a typical simulation setup listed in Table 1, the peak power output was 373 W at nominal Tip Speed Ratio (TSR) of 0.75. This is a 219%

Table 1. Typical simulation setup to verify hypothesis.

No.	Description	Value or Setting
1.	Wind speed	12 m/s
2.	Reynolds number (based on turbine $\varnothing$, 25°C air)	1.709×10^6
3.	Bluff body step height	5.0 m
4.	Turbine half-width	1.0 m
5.	Turbine radius	1.1 m
6.	Distance of axis behind front edge of bluff body	3.0 m
7.	Distance of axis above top surface of bluff body	1.6 m

Power output: 373 W @ TSR 0.75

Figure 1. Turbine installed above bluff body.

improvement compared to a turbine installed above a flat surface, which produced 117 W at TSR of 0.45. The two installation conditions are shown in Figure 1 and Figure 2 respectively.

Power output: 117 W @ TSR 0.45

Figure 2. Turbine installed above flat surface.

1.4 THESIS OUTLINE

The thesis is divided into seven chapters which are briefly described below.

- 1) Introduction. This chapter gives a brief background on wind energy and the motivation for the present research. This is followed by a statement of the research objectives and a summary of the initial preliminary simulations that confirm the hypothesis on which this research is based.
- 2) Literature Review. This chapter reviews the current trends in urban wind energy, the two major classes of wind turbines, namely, axial and cross flow and the sub-classes, namely lift- and drag-based. A review of prior art in flow modification for cross flow turbines and the turbulence models in CFD relevant to the present research is also made.
- 3) Research Methodology. This chapter gives an overall picture of the research methodology to accomplish the research objectives including how experimental work and simulation complement each other.

4) Experimental Work & Discussion of Results. This chapter presents the three stages of experimental work with sufficient detail for future researchers to repeat or extend upon the present research. Experimental results are also presented.

5) CFD Simulations. This chapter describes both the preliminary simulations in support of the design of experimental apparatus as well as the main parametric simulation study used to determine the line of optimal installation position. The usual validation and independence studies are also described.

6) Empirical Model. In this chapter, an empirical formula for the line of optimal turbine installation position is derived from simulation results and the results are analysed to explain the fundamental basis for the nature of the formula.

7) Conclusion & Future Work. In this chapter, the main outcomes of the present research and the activities performed are summarised. After that, recommendations for future work and the practical points to note for future driving tests are presented.

CHAPTER 2. LITERATURE REVIEW

2.1 WIND: A VIABLE SOURCE OF RENEWABLE ENERGY

Among the various sources of renewable electricity sources, wind energy is the second largest source after hydroelectricity, contributing 459.9 TWh in 2011[3]. However, unlike hydroelectricity which is relatively mature and slow growing averaging 3.1% growth per year in the decade up to 2011, wind power in 2011 grew by 30.9% compared to 2010 [3]. As at the end of 2011, global installed wind power capacity was 238 GW [3]. With this rapid growth, equipment and operating costs have declined. Together with improved efficiency, the best performing wind farms have achieved grid parity, while the average performers will do so by 2016 [4].

2.2 URBAN WIND ENERGY

Urban wind energy is receiving increasing interest in recent years. An example of efforts in this area is the Wineur Project [5] to study wind energy integration in the urban environment. Academic interest in this area is also strong. Examples of research include the design of a diffuser for a rooftop turbine by van Beveren [6] , feasibility study of a wind turbine in a farm building rooftop by Bos [2], the study of rooftop wind climate by van Wijk [7], etc. However, urban wind should only be seen as a complementary energy source to large scale rural wind turbines and other renewable energy sources such as solar. One of the reasons is that cost of energy from urban wind is too high to be competitive with fossil fuels. Fortunately, electricity produced by such turbines only have to be cost competitive with retail electricity prices, which are about five times the price of wholesale electricity produced by large turbines [5], since they are available at the point of consumption.

2.3 DRAG- & LIFT-BASED CROSS-FLOW WIND TURBINES

Wind turbines are broadly classified into cross flow and axial flow. In cross flow turbines, the direction of wind is perpendicular to the axis of rotation of the turbine. In axial flow turbines, the direction of wind is parallel to the axis of rotation of the turbine. Use of the terms “horizontal axis wind turbine” or HAWT, and “vertical axis wind turbine” or VAWT,

Figure 3. Left: horizontal axis cross-flow wind turbine [8]. Right: conventional vertical axis cross-flow wind turbine[9].

Figure 4. Solar updraft tower in Manzanares, Spain. Inset: vertical axis axial flow wind turbine inside the base of the tower[10].

have deliberately been avoided because they do not describe the fundamental operating principle. Figure 3 illustrates a pair of cross-flow but horizontal axis wind turbines, which have the same operating principle as a conventional VAWT shown on its right. In contrast, Figure 4 shows an axial flow vertical axis turbine (inside the base of the tower, shown inset) which has the same operating principle as a conventional HAWT. Cross flow

Figure 5. Lift-based cross-flow turbines: (left to right) World's largest cross-flow wind turbine, the Éole at Cap Chat, Quebec, built according to the original Darrieus design [11], giromill or H-rotor [9], Pison Cycloturbine with phase-dependent and wind-speed-dependent pitch variation [12], helical version of giromill [13].

Figure 6. Drag-based cross-flow wind turbines: (clockwise from top left) Three-bladed Savonius used in the present research, helical version of Savonius [14], cup-anemometer-style turbine [15], Ecowing with phase-dependent blade pitch variation[16].

Figure 7. Combined drag- and lift-based cross-flow wind turbines: (left to right) Hi-VAWT [17], Lenz turbine[18], Cygnus Power [19].

turbines are further classified into lift-based, drag-based and combination of lift- and drag-based types. A representative selection of each class is shown in Figure 5, Figure 6 and Figure 7 respectively.

The efficiency of a wind turbine is expressed as its coefficient of power c_p . It is the ratio of turbine power output to the power of the wind. Consider a tube of wind with cross sectional area equal to the frontal swept area of a wind turbine. As this tube of wind passes through the turbine, part of its energy is extracted by the turbine. It is impossible to extract all the energy, because that would mean that all the kinetic energy in the wind is removed and the wind becomes still air, unable to move away after passing through

Figure 8. Control volume for Betz's derivation.

the turbine. Betz [20] showed that the maximum fraction of the wind power that can be extracted by a wind turbine is 0.593. To summarise Betz's derivation, consider a control volume enclosing a stream tube with no flow across the assumed conical surface, as shown in Figure 8. Flow enters only through the smaller circular cross section of the control volume and, since the wind speed decreases after extraction of some of its energy, exits only through the larger circular cross section. Near the middle of this control volume is an actuator disc which represents the fluid turbine. On each side of the actuator disc, the Bernoulli's equation can be written:

$$p_1 + \frac{1}{2}\rho U_1^2 = p_2 + \frac{1}{2}\rho U_2^2 \quad (2.1)$$

$$p_3 + \frac{1}{2}\rho U_3^2 = p_4 + \frac{1}{2}\rho U_4^2 \quad (2.2)$$

The thrust on the actuator disc is obtained by considering the pressure drop across it:

$$T = A(p_2 - p_3) \quad (2.3)$$

The static pressure far upstream and far downstream of the actuator disc are equal to the ambient static pressure ($p_1 = p_4$) and wind speed across the disc remains the same ($U_2 = U_3$). Hence combining equations 2.1 to 2.3, we have:

$$T = \frac{1}{2}\rho A(U_1^2 - U_4^2) \quad (2.4)$$

Applying the conservation of momentum to the one-dimensional incompressible steady flow, the thrust on the actuator disc can also be expressed as:

$$T = \rho A U_2 (U_1 - U_4) \quad (2.5)$$

Combining equations 2.4 and 2.5,

$$U_2 = \frac{U_1 + U_4}{2} \quad (2.5)$$

Equation 2.5 means that the velocity at the rotor is the average of the upstream and downstream wind speeds. Defining the induction factor as the fractional change in wind speed from the far upstream to the face of the actuator disc:

$$a = \frac{U_1 - U_2}{U_1} \quad (2.6)$$

and expressing the other wind speeds in terms of a :

$$U_2 = U_1(1 - a) \quad (2.7)$$

$$U_4 = U_1(1 - 2a) \quad (2.8)$$

Power output of the actuator disc is given by the thrust on the disc multiplied by the velocity at the disc, substituting into equation 2.5:

$$P = TU_2 = \rho AU_2^2(U_1 - U_4) \quad (2.9)$$

Substituting equations 2.7 and 2.8:

$$P = \rho AU_1^3(1 - a)^2 2a \quad (2.10)$$

The coefficient of power is the power output of the actuator disc expressed as a fraction of the wind power. Hence:

$$c_P = \frac{P_{actuator}}{P_{wind}} = \frac{\rho AU_1^3(1-a)^2 2a}{\frac{1}{2}\rho AU_1^3} = 4a(1 - a)^2 \quad (2.11)$$

To find maximum coefficient of power, differentiate equation 2.11 with respect to a ,

$$\frac{dc_P}{da} = 4a[2(1 - a)(-1)] + (1 - a)^2 4 = 12a^2 - 16a + 4 \quad (2.12)$$

and find the roots of the derivative,

$$0 = 12a^2 - 16a + 4 \quad (2.13)$$

which gives us the non-trivial root of $a = 1/3$. At this value of a :

$$c_{P,max} = 4 \left(\frac{1}{3}\right) \left(1 - \frac{1}{3}\right)^2 = \frac{16}{27} = 0.593 \quad (2.14)$$

In contrast, the maximum coefficient of power for a drag based machine does not exceed 0.35, depending on the coefficient of drag c_D of the blade that is employed, as shown in the following derivation:

Consider a generic blade with frontal area A moving linearly in the same direction as the wind velocity U_1 , but at a slower magnitude U_2 , as shown in Figure 9. The force on the blade F_D and hence the power produced by the blade are:

$$F_D = c_D \left[\frac{1}{2} \rho A (U_1 - U_2)^2 \right] \quad (2.15)$$

$$P = c_D \left[\frac{1}{2} \rho A (U_1 - U_2)^2 \right] U_2 \quad (2.16)$$

Consider the case where the blade is stationary. The relative speed of the wind striking the blade ($U_1 - U_2$) is at its maximum and thus the force F_D on the blade is the maximum. However, since the blade is stationary, no power is produced. Consider the other extreme case where the blade is moving at the speed of the wind. Although the blade speed is at the maximum possible, there is no relative movement between the blade and the wind and hence zero drag. Thus the power is again zero. It follows that

Figure 9. Schematic of a drag-driven blade.

power output must be dependent on the relative speed of the blade with respect to the wind speed and there must exist a blade speed between the two extremes where the power output is maximum. With this reasoning, we define the speed ratio of the blade to the far upstream wind speed, also known as the Tip Speed Ratio (TSR) for the case of a rotating blade:

$$\lambda = U_2/U_1 \quad (2.17)$$

The power expressed in terms λ of is:

$$P = c_D \frac{1}{2} \rho A (U_1 - \lambda U_1)^2 \lambda U_1 = c_D \frac{1}{2} \rho A U_1^3 \lambda (1 - \lambda)^2 \quad (2.18)$$

The coefficient of power is thus:

$$c_p = \frac{P_{actuator}}{P_{wind}} = c_D \frac{\frac{1}{2} \rho A U_1^3 \lambda (1 - \lambda)^2}{\frac{1}{2} \rho A U_1^3} = c_D (1 - \lambda)^2 \lambda \quad (2.19)$$

Once again, to find the maximum c_p we differentiate it with respect to λ and solve for the roots:

$$\frac{dc_p}{d\lambda} = c_D (1 - 4\lambda + 3\lambda^2) = 0 \quad (2.20)$$

giving the non-trivial solution of $\lambda = 1/3$. At this value of λ :

$$c_{p,max} = c_D \left(1 - \frac{1}{3}\right)^2 \frac{1}{3} = \frac{4}{27} c_D = 0.148 c_D \quad (2.21)$$

Thus we see that even with the ideal situation of a turbine having an open semi-cylindrical blade of infinite length at the instant when the opening faces the wind perpendicularly and the advancing blade completely shielded from the wind, the c_D is 2.3 [21] thus giving an instantaneous $c_{p,max}$ of only 0.341. For the case of a flat plate with c_D of 2 [21], the $c_{p,max}$ drops to 0.296.

The above theoretical limitations are seen in practical devices. A well optimised lift-based turbine such as the Pinson Cycloturbine has a peak c_p of 0.4 [22]. In contrast, c_p

for Savonius (drag-based) devices does not exceed 0.2 [23]. However, when selecting a turbine for deployment, one does not merely make a decision on the basis of its c_p . For example, Blackwell, *et al.* [24] mentions that the less efficient Savonius turbine may find more applications in developing countries. Bos [2], when assessing the feasibility of the WindNok drag-based turbine, uses a weighted objectives method that includes suitability for Do-It-Yourself installation and likelihood of approval from authorities. Oy Windside Production Ltd. [14] uses a helical Savonius design as the basis for their highly durable products intended for use in harsh remote environments such as Antarctica where its simplicity, low cut-in wind speed, high survival wind speed are overriding factors.

In a drag-based turbine, useful torque is produced by a blade only if the air flow has the same direction as the blade motion but is of a higher speed. Therefore the ideal airflow condition to power a drag-based turbine is a circular flow in the same direction of rotation as the turbine, but at a higher angular speed than the turbine's rotation. This kind of flow condition is conveniently found above a step-shaped bluff body that is subjected to wind, e.g., a building in the shape of a cuboid. Therefore, in the present research, in order to exploit the circulating flow above a bluff body the drag-based Savonius turbine has to be used.

2.4 AIRFLOW MODIFICATIONS FOR CROSS-FLOW TURBINES

There have been numerous attempts to improve the efficiency of cross-flow turbine by some form of flow modification. These devices include ducts, diffusers, inlet guide vanes, etc., examples of which are shown in Figure 10. Other characteristics such as cut-in wind speed may also be improved by these devices. Park *et al.* [25] and Takao *et al.* [26] have conducted tests of cross-flow wind turbines fitted with guide vanes. Park reported that the c_p is 0.47, which is very impressive for a drag-based design. Takao reported that when a conventional three-bladed giromill was fitted with guide vanes, the c_p improved to 0.2, up from 0.12 for the same turbine without guide vanes. These two works utilise the concept of additional elements, which adds to the cost and complexity of the turbine. Of greater simplicity is to find common buildings that have already been built for other purposes and use them as flow modifiers. Bos [2] carried out

a feasibility study on integrating wind turbines to the pitched (inclined) roof of a farm building. The type of turbine selected was Savonius. He concluded that the concept was poorer than installing photovoltaic panels on the roof. Van Beveren [6] utilised both the building and additional diffuser plates to improve the performance of helical giromill. The system improves the peak coefficient of power from 0.12 to 0.16. Wind speed was also increased such that the turbine produces 40 – 80% more power.

Mohamed [27] *et al.* attempted to optimise two- and three-bladed Savonius turbines using a simple inclined plate to shield the returning blade from the wind. The position and angle of the plate was optimised by an Evolutionary Algorithm coupled with ANSYS Fluent using a two-dimensional domain. The optimal plate positions improves the coefficient of power from 0.18 to 0.25 for the two-bladed case and from 0.15 to 0.21 for the three-bladed case.

Figure 10. Airflow modification: (clockwise from top left) ducted Honeywell Turbine [28], Donqi diffuser augmented turbine, KR Windpower cross-flow turbine with guide vanes [29], horizontally mounted version of the KR Windpower turbine manufactured by VQ Wind [30].

Altan [31] *et al.* further expanded the above technique by using a funnel comprising two rectangular and two sector-shaped plates which direct air into the advancing side of the turbine. Various funnel sizes and angles were tested and the optimal arrangement improves the coefficient of power from 0.16 to 0.39.

Among these prior works, none of them utilise the concept of flow over a sharp-edged building as a flow modification to drive a drag-based Savonius turbine, which is expected to benefit most from the circular flow pattern.

2.5 TURBULENCE

Turbulence is a condition in fluid flow where the velocity, pressure and other flow properties at given points in the turbulent region change stochastically with time. The Reynolds number (Re) of a flow, defined as:

$$Re = \frac{\rho UL}{\mu} \quad (2.22)$$

where: ρ : density of the fluid
 U : characteristic velocity of the flow
 L : characteristic length of the flow
 μ : dynamic viscosity of the fluid

determines whether it is turbulent, with the onset of turbulent flow marked by a value of Re higher than some critical value. Due to the randomness of turbulent flow, it is not easily represented by simple formulae. The common practice in CFD is to decompose the flow properties, represented generally by $\varphi(t)$, into a steady mean value Φ with a fluctuating component $\varphi'(t)$ superimposed on it, a technique known as Reynolds decomposition:

$$\varphi(t) = \Phi + \varphi'(t) \quad (2.23)$$

Consider the instantaneous continuity and Navier-Stokes momentum equations with velocity vector $\mathbf{u}$, comprising the Cartesian components u , v and w :

$$\text{div } \mathbf{u} = 0 \quad (2.24)$$

$$\frac{\partial u}{\partial t} + \text{div}(u\mathbf{u}) = -\frac{1}{\rho} \frac{\partial P}{\partial x} + \nu \text{div}(\text{grad } u) \quad (2.25a)$$

$$\frac{\partial v}{\partial t} + \text{div}(v\mathbf{u}) = -\frac{1}{\rho} \frac{\partial P}{\partial y} + \nu \text{div}(\text{grad } v) \quad (2.25b)$$

$$\frac{\partial w}{\partial t} + \text{div}(w\mathbf{u}) = -\frac{1}{\rho} \frac{\partial P}{\partial z} + \nu \text{div}(\text{grad } w) \quad (2.25c)$$

Applying Reynolds decomposition (eq. 2.23) to the velocity vector $\mathbf{u}$ and its components and pressure P :

$$\mathbf{u} = \mathbf{U} + \mathbf{u}', u = U + u', v = V + v', w = W + w' \text{ and } p = P + p'$$

From eq. 2.24, for the case of an incompressible flow,

$$\text{div } \mathbf{U} = 0 \quad (2.26)$$

Applying Reynolds decomposition to the momentum equations and re-arranging so that the Reynolds stress terms are grouped together:

$$\frac{\partial U}{\partial t} + \text{div}(U\mathbf{U}) = -\frac{1}{\rho} \frac{\partial P}{\partial x} + \nu \text{div}(\text{grad } U) + \frac{1}{\rho} \left[\frac{\partial(-\rho \overline{u'^2})}{\partial x} + \frac{\partial(-\rho \overline{u'v'})}{\partial y} + \frac{\partial(-\rho \overline{u'w'})}{\partial z} \right] \quad (2.27a)$$

$$\frac{\partial V}{\partial t} + \text{div}(V\mathbf{U}) = -\frac{1}{\rho} \frac{\partial P}{\partial y} + \nu \text{div}(\text{grad } V) + \frac{1}{\rho} \left[\frac{\partial(-\rho \overline{u'v'})}{\partial x} + \frac{\partial(-\rho \overline{v'^2})}{\partial y} + \frac{\partial(-\rho \overline{v'w'})}{\partial z} \right] \quad (2.27b)$$

$$\frac{\partial W}{\partial t} + \text{div}(W\mathbf{U}) = -\frac{1}{\rho} \frac{\partial P}{\partial z} + \nu \text{div}(\text{grad } W) + \frac{1}{\rho} \left[\frac{\partial(-\rho \overline{u'w'})}{\partial x} + \frac{\partial(-\rho \overline{v'w'})}{\partial y} + \frac{\partial(-\rho \overline{w'^2})}{\partial z} \right] \quad (2.27c)$$

The above four equations are known as the Reynolds-averaged Navier-Stokes (RANS) equations. Equations for time-averaged scalar properties with similar extra turbulent terms can be derived in the same manner. These additional terms are then modelled with turbulence models. Equations for compressible flow are slightly different due to the need to consider non-constant density. However, since the present research involves only incompressible flow, they are not elaborated here. To handle the Reynolds stresses, Boussinesq proposed the use of turbulent viscosity or eddy viscosity μ_t in the usual Newtonian viscosity equation using the mean flow velocities:

$$\tau_{ij} = -\rho \overline{u'_i u'_j} = \mu_t \left(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right) \quad (2.28)$$

where i and j represent two perpendicular Cartesian directions.

Among the many turbulence models available, the k - ϵ model is the most widely used and validated model [32]. In this model the turbulent kinetic energy k and the rate of dissipation of turbulent kinetic energy ϵ are used to obtain the eddy viscosity μ_t as follows:

$$\mu_t = \rho C_\mu \frac{k^2}{\epsilon} \quad (2.29)$$

The two equations for solving k and ε are:

$$\frac{\partial(\rho k)}{\partial t} + \text{div}(\rho k \mathbf{U}) = \text{div}\left(\frac{\mu_t}{\sigma_k} \text{grad } k\right) + 2\mu_t S_{ij} \cdot S_{ij} - \rho \varepsilon \quad (2.30)$$

$$\frac{\partial(\rho \varepsilon)}{\partial t} + \text{div}(\rho \varepsilon \mathbf{U}) = \text{div}\left(\frac{\mu_t}{\sigma_\varepsilon} \text{grad } \varepsilon\right) + C_{1\varepsilon} \frac{\varepsilon}{k} 2\mu_t S_{ij} \cdot S_{ij} - C_{2\varepsilon} \rho \frac{\varepsilon^2}{k} \quad (2.31)$$

where $S_{ij} \cdot S_{ij}$ is the dot product of the rate of deformation tensor of the mean flow. The five constants in eq. 2.29 to eq. 2.31 are obtained by fitting data from a wide range of turbulent flows as follows:

$$C_\mu = 0.09; \quad \sigma_k = 1.00; \quad \sigma_\varepsilon = 1.30; \quad C_{1\varepsilon} = 1.44; \quad C_{2\varepsilon} = 1.92$$

Summing the normal Reynolds stress terms in eq. 2.27a - c, it is found that it is exactly equal to twice the negative of turbulent kinetic energy per unit volume: $-\rho(\overline{u'^2} + \overline{v'^2} + \overline{w'^2}) = -2\rho k$. Hence assuming isotropic behaviour of the normal Reynolds stresses, each of them is given a value of $(2\rho k)/3$. However, it is to be noted that this assumption is not always accurate. This method is expressed in the modified Boussinesq expression:

$$\tau_{ij} = -\rho \overline{u'_i u'_j} = \mu_t \left(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right) - \frac{2}{3} \rho k \delta_{ij} \quad (2.32)$$

where δ_{ij} , the Kronecker delta, is 1 if $i = j$ and zero if $i \neq j$.

Although the k - ε model is widely used, it does not perform well in unconfined flows, flows with large strains (such as those over curved surfaces), rotating flows and flows in non-circular ducts. As reported by Murakami [33] *et al.*, its key shortcoming arises from the inability of the term for turbulence kinetic energy production to take on negative values, thus over-predicting the turbulence level in front of a forward facing step bluff body. Due to this over-prediction, the observed re-circulating flow above the bluff body is also not reproduced by the model. Hence, the k - ε model is not recommended for flows with separation. For the present research involving a cross flow turbine, the blades undergo full circles of angle of attack variation during normal operation, hence flow separation is inevitable. A technique for reducing the problem of excessive turbulence prediction according to Durbin and Pettersson Reif [34] is to impose bounds on the turbulent viscosity, for example, by limiting it to:

$$\mu_t = \min\left(\frac{C_\mu k^2}{\varepsilon}, \frac{\alpha k}{|S|}\right) \quad (2.33)$$

with $\alpha \leq 1/\sqrt{6}$.

For the flows encountered in the present research, the Shear Stress Transport (SST) k - ω model of Menter [35] is more appropriate. This model arose from the original k - ω model of Wilcox [36]. In this model, the turbulence frequency ω (originally named the specific dissipation rate by Wilcox) is defined as:

$$\omega = \varepsilon/k \quad (2.34)$$

Thus the eddy viscosity is:

$$\mu_t = \rho k/\omega \quad (2.35)$$

The transport equations for solving k and ω are:

$$\frac{\partial(\rho k)}{\partial t} + \text{div}(\rho k \mathbf{U}) = \text{div} \left[\left(\mu + \frac{\mu_t}{\sigma_k} \right) \text{grad } k \right] + P_k - \beta^* \rho k \omega \quad (2.36)$$

where $P_k = 2\mu_t S_{ij} \cdot S_{ij} - \frac{2}{3}\rho k \frac{\partial U_i}{\partial x_j} \delta_{ij}$ is the rate of production of turbulent kinetic energy, and $S_{ij} \cdot S_{ij}$ is the dot product of the rate of deformation tensor of the mean flow.

$$\begin{aligned} \frac{\partial(\rho \omega)}{\partial t} + \text{div}(\rho \omega \mathbf{U}) = & \text{div} \left[\left(\mu + \frac{\mu_t}{\sigma_\omega} \right) \text{grad } \omega \right] \\ & + \gamma_1 \left(2\rho S_{ij} \cdot S_{ij} - \frac{2}{3}\rho \omega \frac{\partial U_i}{\partial x_j} \delta_{ij} \right) - \beta_1 \rho \omega^2 \end{aligned} \quad (2.37)$$

However, the model is very dependent on the proper setting of ω at the free stream boundary condition. Thus Menter developed the SST model. The model uses the standard k - ε model (which is less sensitive to assumed values in the free stream) in the fully turbulent region far from the wall and transformation into a k - ω model in the near wall region. The k -equation is unchanged from Wilcox's model but the transformed ω -equation is as follows:

$$\begin{aligned} \frac{\partial(\rho \omega)}{\partial t} + \text{div}(\rho \omega \mathbf{U}) = & \text{div} \left[\left(\mu + \frac{\mu_t}{\sigma_{\omega,1}} \right) \text{grad } \omega \right] \\ & + \gamma_2 \left(2\rho S_{ij} \cdot S_{ij} - \frac{2}{3}\rho \omega \frac{\partial U_i}{\partial x_j} \delta_{ij} \right) - \beta_2 \rho \omega^2 + 2 \frac{\rho}{\sigma_{\omega,2} \omega} \frac{\partial k}{\partial x_k} \frac{\partial \omega}{\partial x_k} \end{aligned} \quad (2.38)$$

To limit over prediction of shear stress in adverse pressure gradient boundary layers, limiters on the eddy viscosity μ_t and rate of production of turbulent kinetic energy P_k are employed. Blending functions give a smooth transition between the far field and near

wall. More recently, Menter improved the SST model with revised model constants as follows:

$$\sigma_k = 1.0; \quad \sigma_{\omega_1} = 2.0; \quad \sigma_{\omega_2} = 1.17; \quad \gamma_2 = 0.44; \quad \beta_2 = 0.083; \quad \beta^* = 0.09;$$

Additionally, CFX also incorporates the curvature correction technique of Spalart and Shur [37], thus making the solver more suitable for the present research on a rotating turbine.

CHAPTER 3. RESEARCH METHODOLOGY

3.1 OVERVIEW OF METHODOLOGY

The objective of the present research is to determine the line of optimum installation position for a turbine above a forward facing step bluff body. The result for one combination of parameters, comprising step height, wind speed and turbine size, will be determined, but the methodology will be useful for determining results for other combinations of parameters. The following sections explain the methodology to achieve this objective. Figure 11 summarises the proposed methodology for the present research, the final product of which will be a body of knowledge useful for future users planning to install such a turbine on top of a building.

3.2 EXPERIMENTAL WORK

The author is fortunate to have funding from the Maritime and Port Authority of Singapore for the present research. The funding enabled experiments to be performed to obtain data for validating CFD simulation. The main basis for the experiments was driving tests of the turbine rig on a vehicle. Preliminary experiments conducted in the NTU closed circuit wind tunnel revealed two problems: 1) the scale of the apparatus is very much smaller than a full size turbine designed to produce practical amounts of power, and 2) the torque produced was very low, close to the resolution of the torque sensor and also comparable to sources of error such as the natural springiness of the connecting wires of the brake. These are the reasons for the adoption of driving tests. Nevertheless, wind tunnel experiments were used to support design decisions on the driving test rig.

3.2.1 WIND TUNNEL TESTS

To determine the installation position of the turbine on the driving test rig, wind tunnel testing was used, in spite of its drawbacks. First of all, tests were carried out to determine the required forward facing step to create a flow field with a separation bubble above the step and gradually decreasing to the desired free stream wind speed at the ceiling of the test section. Once this has been done, testing with a reduced-scale

Figure 11. Flowchart of methodology.

wind turbine to find the installation position for maximum output can be carried out. In view of the difficulty in obtaining meaningful torque measurements, the torque was fixed at a practical and convenient value. The position of the turbine was then varied and its rotating speed measured. The point of highest speed was deemed to be the optimal position. The position was normalised with the turbine diameter. Results obtained from this stage of work were used to determine the installation position of the driving test turbine, described in the next section.

3.2.2 DRIVING TEST OF TURBINE TEST RIG

The driving test rig consists of a bluff body and a turbine installed above it at a position most likely to produce maximum power. The rig was secured safely onto the load bed of a lorry. The lorry was driven at test speeds of 8, 10 and 12 m/s. Loading of the turbine was performed by a mechanical disk brake actuated by a motor driven by a digital pulse-width modulated (PWM) electrical supply. Instruments include optical sensors to measure turbine and lorry speed, load cell to measure braking torque and an anemometer to measure wind speed at a practicable position beside the turbine. The lorry speed is also displayed on a scrolling digital display with resolution of 0.1 m/s to assist the driver in maintaining the desired speed. Data was recorded on a PC-based oscilloscope. The test site was a 2.4 km long straight section of Lim Chu Kang Road.

3.2.3 CHOICE OF TEST PERIOD AND TIME

October was selected as the time of the year for conducting the experiments. The main consideration is that it is inter-monsoon. Figure 12 shows the average monthly wind speed in Singapore, showing that October and November are the months of lowest prevailing wind speed. Additionally, October has a lower average number of rainy days than November [38]. Figure 13 shows the hourly wind speed for the inter-monsoon month of April 2012 for Pandan Gardens (the nearest location to the test site for which data is available), showing that the early morning hours have the lowest wind speeds [39]. Therefore experiments were conducted from 2 am to 5 am in the morning, which coincidentally minimises disruption to public traffic.

Month	Direction	Mean speed (m/s)	Max gust speed (m/s)
January	N/NE	2.6	20.3
February	N/NE	2.8	17.8
March	N/NE	2.1	21.9
April	Variable	1.4	23.9
May	S/SE	1.5	18.1
June	S/SE	1.9	21.4
July	S/SE	2.3	23.9
August	S/SE	2.4	21.9
September	S/SE	1.9	21.4
October	Variable	1.4	20.3
November	Variable	1.3	21.1
December	N/NE	1.9	17.2

Figure 12. Average wind monthly wind speeds [38].

Figure 13. Diurnal mean wind speed for Pandan Gardens in April 2012 [39].

3.3 COMPUTATIONAL FLUID DYNAMICS SIMULATION

3.3.1 SOFTWARE

In view of the multitude of parameters to be studied, it would be impossible to perform experiments on all possible combinations of parameters. Hence the primary means of obtaining results was CFD simulation. A commercial software, ANSYS CFX in the Workbench environment was used. CFX uses the finite volume method. The simulation domain is discretised using a mesh. The centres of adjacent mesh elements define the control volume and thus the solution variables and fluid properties are calculated for the

vertices of the mesh. CFX is an implicit pressure-based coupled solver. Throughout the present research the simulations involving air assumes that it has constant properties at 25°C, taken from the ANSYS materials library. Among the many capabilities of CFX, its transient flow and rotating sub-domain capabilities was used. It also offers a wide selection of turbulence models and transient discretisation schemes. The Workbench environment provides user-friendly pre- and post-processing capabilities. Furthermore, integration with the Computer Aided Design (CAD) model allows physical dimensions such as length, position, etc., to be set as parameters that can be varied and studied. Other parameters such as wind speed and TSR can similarly be studied.

Figure 14. The Workbench Environment.

Figure 14 is a screen capture of the ANSYS Workbench window. In the main panel, a column of boxes list the sequential steps that have to be performed to carry out a CFD simulation. The first step is the reading of the geometry. A wide range of file formats is compatible. In this case, a native SolidWorks CAD file is used. The practice preferred by the author is to create the fluid domains, i.e., any solids in the domain such as the

turbine blades are voids. This eliminates the need to mesh the often thin solid sections. The second step, meshing, is performed by the ANSYS Meshing software. Meshing is largely automated, thus even complex geometries can be meshed with minimal human intervention. At the same time, a wide range of controls are available to optimise the mesh, for example, increasing fineness at selected regions. Mesh fineness can also be parametrically varied and the resulting specified output parameters such as y^+ or Courant number can be monitored. The mesh quality is automatically checked after meshing and the mesh file will only be written if quality is acceptable. The third step is the setting up of the simulation itself, by defining the transient simulation settings, the domain properties, the boundary conditions and result saving settings. The next step is starting and monitoring the CFX solver. After the solver has completed its run, the results are processed with CFD-Post, which can present them graphically or extract numerical data for further processing with other software such as Microsoft Excel.

3.3.2 VALIDATION

Whenever a turbulence model is employed in CFD simulation, there is a risk that the simulation results may not be identical to reality. Therefore it is important to have experimental data to validate simulations. This is particularly necessary in simulations of cross flow turbines, where the local Reynolds number experienced by the turbine blades changes cyclically over each revolution. Hence applicability of a particular turbulence model has to be validated. CFX is well validated for problems similar to the present research. The most relevant example being the work of McLaren *et al* [40], who performed CFD simulation of a high solidity Vertical Axis Wind Turbine (VAWT) and compared the result with experimental measurements of wake velocities, flow visualisation and power coefficient. Good agreement was obtained when the Shear Stress Transport (SST) turbulence model was employed. Another example is the work of Nobile *et al* [41], who performed simulation of a typical low solidity VAWT and found good agreement of vorticity field with experimental data, again using the SST model.

Notwithstanding the proven predictive capability of CFX, results from experiments described in section 3.2 were used to validate simulation of the experimental setup. Adjustments of parameters such as mesh fineness, domain size, simulated duration, etc.,

were made. Only when agreement between experimental and simulation result is reached can simulation results of further cases be accepted with confidence.

3.3.3 INDEPENDENCE STUDIES

In line with good CFD simulation practice, independence studies were performed together with the validation process. Grid independence study is mandatory. Additionally in the present research, due to the presence of a bluff body in the domain, ensuring independence of computational domain size is necessary. It has to be proven that the chosen domain size gives identical results as any larger domain.

Figure 15. Torque obtained by simulation, clearly showing that for the case with bluff body (BB), settling occurred much later.

Another characteristic of the simulations in the present research is the need for a sufficiently long simulated duration in order for the torque result to settle into a constant amplitude periodic pattern. For example, in Figure 15, obtained during the preliminary verification of hypothesis, the blue plot shows the torque vs. time for a turbine installed above an ordinary flat surface, while the red plot shows the case

installed above a bluff body. It can be seen that for the former, settling occurs after about three periods (one revolution) while for the latter, settling occurred after 24 periods (eight revolutions). Therefore, studies will be performed to ensure that the simulated duration is sufficient to obtain a settled torque result.

3.3.4 PARAMETRIC STUDIES AND NORMALISATION

The work described in the foregoing sections set the stage for CFD work proper, namely, to find the turbine performance under different operating and installation conditions. The parameters that were studied are installation position relative to the edge of the bluff body and TSR as illustrated by Figure 16. Normalisation of position using both the step height and the turbine radius were carried out so that the results can be generalised for other step heights.

Figure 16. Parameters that were studied.

Table 2 lists the parameters that are likely to affect the coefficient of power c_p . First of all is the wind speed as the power per unit cross sectional area is proportional to the cube of wind speed. As the wind flows over a forward facing step bluff body, a re-circulating separation bubble forms above the top of the step. The size of the bubble and its re-attachment point is related to step height. Within this bubble, flow velocity ranges from being opposite in direction to the oncoming wind near the surface of the step, to zero at the approximate centre of the bubble, to higher than free stream wind speed at the top periphery of the bubble. The torque of the turbine depends on how the advancing and returning blades interact with different regions in the separation bubble and hence it is reasonable to suspect that both the radius and position of the turbine are

parameters that affect c_p . Additionally, since the size of flow features depend on the step height, it would also be reasonable to normalise these parameters by dividing by step height. Finally, as with all wind turbines, the rotating speed, normalised as TSR, also affects the c_p . Although the non-dimensional form of wind speed Reynolds number is often used as a parameter in bluff body studies, in the present research, its dimensional form in m/s is used throughout.

Table 2. Parameters affecting coefficient of power.

No.	Parameter	Non-dimensional form	Varied in the present study?
1.	Wind speed	Reynolds number	No (fixed 12 m/s)
2.	Step height	N.A.	No (fixed 5 m)
3.	Turbine radius	Divide by step height	No (fixed 1.25 m)
4.	Turbine axis horizontal position, x	Divide by step height	Yes
5.	Turbine axis vertical position, y	Divide by step height	Yes
6.	Rotating speed	TSR	Yes

3.3.5 ESTABLISHMENT OF REFERENCE FLOW FIELD

The flow above a forward facing step is a well-studied topic and thus useful as a reference for the description of the optimal position that was discovered in the present research. The lines of minimum, free stream and maximum wind speeds in the separation bubble above the step were extracted and compared against the line of optimal installation position. This gives a basis when analysing the underlying mechanisms for the optimal position and also gives future users a convenient surrogate to derive optimal position when detailed results for their particular situation are not available.

3.3.6 COMPARISON WITH THREE BASELINE CASES

Three baseline turbine performances were established using CFD simulation. A summary of the three baselines is illustrated in Figure 17. These baselines offer a basis for comparing the performance of a turbine installed above a bluff body with the null or conventional condition. The first case is the null condition, where the bluff body is absent and the turbine simply installed above a flat surface, with its axis horizontal and its lower half moving against the wind (returning). The second case is for a turbine installed above a flat surface with its lower half moving with the wind (advancing). This is

used to offer insight into whether the null condition is a non-optimal base. The third case is the conventional method of installation of cross flow drag-based turbines, namely, with its axis vertical above a flat surface.

Figure 17. Baselines. Left: null condition, centre: conventional installation, right: to verify whether null condition is non-optimal.

4.1 OVERVIEW OF EXPERIMENTAL WORK

Experimental work consisted of:

- 1) wind tunnel tests to determine the required wind tunnel apparatus to generate the flow field of a forward facing step usable for subsequent test of a reduced-scale turbine,
- 2) wind tunnel tests to determine the optimum installation of a turbine on the driving test rig, and,
- 3) driving tests of a metre-sized turbine installed above a bluff body using a lorry.

Figure 18. Three parts of experimental work, (left to right) wind tunnel, wind tunnel with reduced-scale turbine, driving test.

A first assessment of how flow over a forward facing step can improve the wind turbine performance was performed in the wind tunnel. A concern with the wind tunnel tests was the vast difference in scale of the wind tunnel turbine model and any actual turbine. Hence, CFD simulation was also used to confirm the optimum installation position that had been obtained by wind tunnel tests. These simulations are explained in Chapter 5.

Results from the driving tests were used to validate CFD simulations of the driving test set up. Once the CFD results have been validated, similar simulation settings are used to perform parametric studies to develop an empirical model of turbine performance with varying installation and operating conditions.

An additional benefit of the experimental work, especially the driving test, was that practical knowledge about such tests was acquired. These dos and don'ts are described in Appendix B and will serve as a useful reference for future researchers in NTU planning to conduct similar tests.

4.2 WIND TUNNEL FACILITY

The wind tunnel facility used in the present research is located in the Main Aircraft Laboratory of School of Mechanical & Aerospace Engineering, NTU. The wind tunnel is an AF6407 closed-circuit wind tunnel with cooling of the circulating air, supplied by STEM-ISI Impianti S.P.A. The system also comes with two closed type test sections, one of them a bare test section and the other with a force balance. The bare test section was used throughout the present research. The internal dimensions of this test section are 0.78 m (W) $\times$ 0.72 m (H) $\times$ 2 m (L). The air velocity inside the test section is continuously variable from 3 m/s to 90 m/s, corresponding to Mach number of 0.029 to 0.26. The air speed distribution inside the test section is constant over about 80% of the test section area. Airflow in the test section has a low turbulence intensity of about 0.1% over a large velocity range due to the provision of the high contraction ratio of 9:1 and a suitable number of turbulence reducing screens. A photograph of the wind tunnel is shown in Figure 19.

Figure 19. Closed-loop wind tunnel.

4.3 WIND TUNNEL TESTS WITH FORWARD FACING STEP

A flow field above a forward facing step is expected to show a separation bubble [42]. Moving vertically upward from the top surface of the step, the wind speed is expected to decrease to zero at the centre of the bubble, then increase to some maximum value and

finally decrease to nearly the desired free stream wind speed at the ceiling of the test section. An insufficient step height would give too small a separation bubble thus requiring an impracticably small turbine to exploit. On the other hand, an excessive step height will cause wind speed near the ceiling to be much higher than free stream, thus deviating from the expected condition in a field installation. This part of the experiment was to find a suitable step height and test section inlet wind speed for subsequent testing with a reduced-scale turbine.

4.3.1 DESIGN AND METHOD

The forward facing step in the test section was constructed using aluminium extruded profiles and acrylic sheets. It extends the full width of the test section, thus forming a

Figure 20. Wind tunnel test of step blockage.

two dimensional obstruction. The step height is adjustable from 0.1 m to 0.5 m. The length of the step is 1 m and installed with its face at the midpoint of the test section. A thermal anemometer, the KIMO VT-200, with measuring range of up to 30 m/s was fitted to the traversing system above the test section to measure the streamwise

component of wind speeds in a grid pattern with good positional repeatability. A range of free stream wind speeds from 2 m/s to 12 m/s was tested.

4.3.2 RESULTS OF WIND TUNNEL STEP TESTS

Figure 21 shows the result that gives the best condition for subsequent testing with a turbine, which is a step height of 0.2 m and free stream wind speed of 8 m/s, thus giving a Reynolds number based on step height of 1.0×10^5 . The maximum streamwise wind speed in the high speed zone of the separation bubble is about 14 m/s and the wind speed near the ceiling drops to about 12 m/s, which is similar to flow over an unconfined forward facing step exposed to 12 m/s free stream wind. Results of CFD simulation of the tested condition (see Section 5.1) is also superimposed for comparison. The error bars show the accuracy of the anemometer, stated as $\pm(3\% + 0.1)$ m/s for wind speed above 3 m/s and $\pm(3\% + 0.03)$ m/s otherwise.

Figure 21. Comparison of simulation and experimental results of step blockage in wind tunnel, wind speed 8 m/s, step height 0.2 m.

4.4 WIND TUNNEL TESTS WITH REDUCED-SCALE TURBINE

4.4.1 DESIGN AND METHOD

To obtain the optimum installation position of the turbine on the driving test rig, a wind tunnel turbine was installed and tested, as shown in Figure 22. A three bladed Savonius turbine 1 with diameter and width both 100 mm was fabricated by the Dimension Elite rapid prototyping (RP) machine. Reynolds number based on wind speed near the ceiling of the test section above the step and turbine diameter was about 7.8×10^4 . The turbine was supported by two flanged instrument bearings installed on two pylons 2, fabricated by the Objet Eden 350V RP machine. The pylons were rotatably attached to their bases 3 which allowed a small amount of height adjustment. On one end of the turbine, a timing pulley and belt 4 transmits the torque to another timing pulley 5 below the horizontal surface of the step blockage. This lower pulley is attached to the shaft of a small DC motor 6 which is energised with selectable duty cycle by a simple microcontroller circuit 7 to load the turbine with a controlled torque. The body of the motor is attached via a rotatable frame to the Chatillon STS-0050 torque sensor 8 for torque measurement. However, when the turbine was tested, the torque obtained was very low, having a maximum value of only 4 mNm at stall. Compared to the torque sensor resolution of 0.5 mNm, this does not give meaningful and reliable results. The torque produced by the

Figure 22. Testing of wind turbine above a step blockage in wind tunnel.

tension in the wires of the motor was comparable in magnitude and hence the mechanical losses in this setup makes it impossible to measure the torque accurately. It was therefore decided to use the turbine RPM at a fixed torque (by shorting the terminals of the DC motor) as an indicator of optimal position. The height of the turbine axis was fixed at 60 mm (60% of turbine diameter) above the top surface of the step. The accuracy of the tachometer is $\pm(1\% + 1 \text{ digit})$ RPM for rotating speed below 6000 RPM, hence error bars are too fine to be displayed in Figure 23.

4.4.2 RESULTS OF WIND TUNNEL TURBINE TESTS

Figure 23 shows the result of wind turbine testing. It can be concluded that to obtain the maximum torque (indicated by maximum turbine speed), the turbine should be located as much forward as possible. Considering the impracticality of having the turbine protrude beyond the front face of the forward facing step, this meant that the position of the turbine axis can only be located at a distance equal to the turbine radius behind the edge of the step.

Figure 23. Wind tunnel test to determine optimum turbine position on the test rig.

4.5 DRIVING TESTS

A search of the current literature shows that vehicle tow testing has been employed by other researchers, such as National Renewable Energy Laboratory (NREL) [43] and Michigan State University [44]. Tow testing offers a practically infinite ‘test section’, which is an advantage found only in field testing. At the same time, it offers accurate and on-demand control of free stream wind speed, which is the advantage in wind tunnel testing. Additionally for the present research, the vehicle can be easily concealed within the bluff body, unlike in conventional turbine testing, where care has to be taken to avoid the wake effects of the vehicle. Furthermore, Singapore is well known for its lack of wind resource, with average wind speeds in the region of 2 m/s, thus making it an ideal location for tow testing because of minimal interference from prevailing wind. Finally, there is a 2.4 km long straight stretch of road sheltered by low hills on each side, just 5 km from the NTU campus, which offers an ideal test site. Thus the decision was made to use driving tests of a large-sized turbine prototype installed on a vehicle as the basis for the experimental methodology for the present research.

4.5.1 DESIGN OF THE SAVONIUS TURBINE

The Savonius turbine used in the present study is a three-bladed device with its axis of rotation installed horizontally. It has a diameter of 1.1 m and width of 1 m. The blades are made of aluminium sheet of thickness 3 mm rolled into a semi-cylinder. Each blade has a flange welded on each end of the semi-cylinder for it to be attached with counter sunk screws to the end plates. The medium density fibre end plates consist of an interior frame shaped like a spoked wheel with circular boards laminated on both sides. The edge is then rounded with a radius equal to its thickness of 19 mm. The round edge faces inwards. Figure 24 is the exploded view of the end plate. The arrangement of the blades and amount of overlap is shown in Figure 25. The overlap of 150 mm (diametrically measured) was adopted with reference from [45]. The choice of this turbine size was made after considering that the largest commercially available Savonius turbines are up to about 2+ m in diameter. Thus testing of a metre-sized turbine does not result in a situation where Reynolds number is excessively different from that of the intended final application. Reynolds number for this turbine based on free stream wind

Figure 24. Exploded view of laminated construction of end plate.

Figure 25. Blade placement, showing curvature and overlap.

speed of 12 m/s and its diameter of 1.1 m is about 8.5×10^5 . In contrast, the Reynolds number of the wind tunnel test turbine in Section 4.4 was a mere 7.8×10^4 . A three-bladed design was chosen in order to reduce torque fluctuation. A consequence of this is the reduction of advantage with the gap near the axis of rotation formed between the overlapped blades. From [24], a two-bladed design gives a maximum c_P of 0.26, while a three-bladed design gives a maximum c_P of only 0.16.

4.5.2 DESIGN OF THE BLUFF BODY

The turbine is installed above a bluff body. The framework of the bluff body is constructed with extruded aluminium profiles. The front face is a sheet of transparent acrylic. The sides are triangular acrylic sheets to allow the lorry doors to open. The top is made of five wooden framed and plywood skinned panels. Cut-outs are made for the vertical supports for the turbine. The bluff body and turbine are carried on board an Isuzu NHR 10-foot lorry. Wire ropes and ratchet tie downs are used to secure it. Figure 26 shows the overall view of the bluff body and turbine installed on the lorry.

Figure 26. Overall view of the driving test lorry with bluff body and turbine.

The choice of bluff body dimensions was made as a result of practical considerations of the vehicle and road dimensions. The frontal face of the driving test rig was not extended fully to the surface of the road because of the need to allow airflow to the radiator as well as to provide clearance for movement of the lorry body on its suspension. The standard lane width on public roads is 2.8 m and together with the standard length of wood and plastic boards being 2.4 m, the choice of 2.4 m for the width of the step was made. The height limit for vehicles is 4.5 m and the present driving test rig has a height of 3.9 m before installing the anemometer and 4.1 m after installing it. Going any higher would compromise the allowance for suspension travel.

4.5.3 POWER TRANSMISSION AND BRAKE

Two shafts, one on each endplate, support the turbine. A chain sprocket on each shaft transmits turbine power to a brake shaft below the top panels of the bluff body. The ratio of the sprockets is 2:1, i.e., the brake shaft rotating speed is twice the turbine rotating speed. Although a more 'high-tech' braking method using a generator was initially considered, in view of the high cost, it was decided to use a mechanical brake instead. Referring to Figure 27, the brake consists of a bicycle brake disc 1 and callipers 2. The callipers are attached to a brake arm 3 that is rotatably installed around the brake shaft 4. A geared DC motor 5 actuates the calliper. Actuation force is controlled by a micro-controller circuit, shown in Figure 32, which is able to send a variable duty-cycle current to the motor to control the braking force. The brake arm is constrained by a load cell 6, the Chatillon SLC-0200 with a range of 0 – 1000 N. Turbine torque is thus given by the product of arm length (0.18 m), the measured force

Figure 27. Braking system.

and the sprocket ratio of two. The brake arm contacts the load cell via a ball castor 7. This ensures that only compressive force is transmitted. To prevent the contact from separating during emergency braking due to chain slack, a spring 8 pulls the ball castor to depress the load cell with some pre-load.

During turbine testing, it was found that in order to obtain fine control of the braking force, the brake disc and brake pads have to be lubricated with SAE140 gear oil. Otherwise, a small change in duty cycle resulted in a large change in braking force, making it impossible to obtain the desired spread of TSRs.

4.5.4 INSTRUMENTATION

The load cell signal (component 6 in Figure 27) is conditioned by the matching Chatillon DFS-R-ND gauge which also has an analogue output. The gauge is automatically zeroed at start-up, thus the pre-load of the load cell is automatically compensated for. An ultrasonic anemometer, the R.M. Young Model 81000, is installed at a practicable

Figure 28. Anemometer position.

position as shown in Figure 28 which also shows the other key dimensions of the bluff body. Figure 29 shows the anemometer installed just prior to driving tests. The anemometer measures the wind speed and it gives the magnitude, elevation, azimuth and air temperature as four analogue signals. The starboard endplate of the turbine is painted with a semi-circular band in contrasting colour, as shown in Figure 26. Turbine speed is measured by an optical sensor, the Optek OPB720B-06Z, sensing the contrasting band. Although an additional sensor was fitted to sense the spokes of the brake disc, it was found that at high turbine speeds, the pulse duration was too short to be captured by available PC oscilloscopes.

Figure 29. Photograph of anemometer as installed for driving test.

The left rear wheel of the lorry is painted with a white semi-circular band, as shown in Figure 30. Wheel speed is also measured by the Optek OPB720B-06Z sensor, sensing the contrasting band. The pulses are also fed into a simple microcontroller circuit which displays the vehicle speed in digital form with a resolution of 0.1 m/s and a scrolling form to assist the driver in maintaining a constant speed. Figure 31 shows the scrolling digital speedometer.

The Hantek 1008A PC oscilloscope connected to a laptop computer is used for recording. Channel 1 is used to record the turbine pulses from the endplate sensor. Channel 2 is used to record the lorry wheel pulses. Channel 3 is used to record the load cell force. Channels 4, 5 and 6 are used to record the wind speed magnitude, elevation and direction respectively.

Figure 30. Wheel speed measurement.

Figure 31. Digital speedometer with scrolling display in metres per second.

Figure 32. Data acquisition by Hantek 1008A PC oscilloscope (centre black box). The DSO-2150 measures brake disc spoke pulses which were not used. The breadboard in the foreground is the brake controller. On the left is the Chatillon force gauge.

Air temperature, being unlikely to fluctuate greatly, is measured by a voltmeter at approximately hourly intervals. As the experiments were conducted between about 2 am to 5 am daily in the inter-monsoon month of October, it was found that temperature was indeed quite constant, ranging between 20.25 to 21.05 degrees Celsius over the two days of driving tests that produced useful results. The general arrangement of the instruments on the lorry dashboard is shown in Figure 32.

4.5.5 LOCATION OF EXPERIMENT

The driving tests were conducted in the western part of Singapore along Lim Chu Kang Road (lat: 1°24', long: 103°42' at its northern end). The road is a straight stretch of about 2.4 km long which offers a three-minute time window for making measurements at 12 m/s. Figure 33 shows a site photograph and satellite view of the test site.

Figure 33. Driving test site: Lim Chu Kang Road. Inset: satellite view, showing a straight section of 2.4 km.

4.5.6 CALIBRATION

The procedure for calibrating the turbine brake load cell is described here. The turbine is rotated by hand such that the edge of a blade is horizontal with the turbine axis and fully braked in that position. A pan for weights is suspended from the edge, as shown in Figure 34. The torque gauge is zeroed. Recording of the analogue output of the gauge is started. A known weight is placed on the pan. A rise in reading of the torque gauge is recorded. The process is repeated for a range of weights from 0.1 kgf to 1.5 kgf. The plot of the result is shown in Figure 35. The intercept of the linear regression of the data points shows an intercept of -0.001237, which converts to 0.5109 N. This is attributed to the friction in the drive train from blade to load cell and will be subtracted from CFD results whenever they need to be compared against experimental results.

The time base of the Hantek PC oscilloscope was also calibrated by supplying square wave of known frequency from a Matrix MFG-8255A function generator, measured with a Tektronix TDS 1012B oscilloscope. The recorded data is processed to obtain the correction for the time base. For calibration of vertical scale, the torque gauge with load cell connected used to supply various voltages to the PC oscilloscope. Weights ranging from 1 kgf to 3 kgf are placed on the load cell and then removed. The supplied voltages

are measured with the Tektronix while the Hantek records data. Recorded data is processed to obtain the scaling in the vertical axis.

Figure 34. Calibration of turbine brake load cell.

Figure 35. Load cell calibration results.

4.5.7 EXPERIMENTAL PROCEDURE

Prior to each day's test, the circumference of the wheel was calibrated using the GPS navigation system consisting of the Endomondo app running on a Sony Xperia Arc S mobile phone. The digital speedometer is programmed with a calibration routine. The lorry is driven between a start point and an end point, typically about 2.4 km. During the drive, the number of revolutions of the lorry wheel is recorded by the speedometer. Using the GPS navigation system, the distance travelled is recorded. Dividing the distance travelled by the number of revolutions gives the wheel circumference which is then used for vehicle speed calculations.

For the load cell reading, in order to record any potential offset and scaling errors in the waveform of the recorded data, before each driving test run, the ball castor contacting the load cell is separated by hand and the resulting negative reading on the gauge recorded. The gauge reading is then used to calibrate the recorded analogue waveform. The prevailing wind magnitude and direction is recorded for 20 seconds and air temperature is recorded.

Figure 36. Test vehicle ready for first test.

The lorry is accelerated to the desired speed. Speed is maintained by the hand throttle. Usually the first run for a particular speed is to obtain the free running speed of the turbine. A typical free running speed at 12 m/s is about 300+ rpm which corresponds to TSR of about 1.5. In subsequent runs, the turbine brake is applied. By trial and error, the amount of braking to achieve a desired TSR is found. The goal is to obtain a range of TSRs from about 0.6 to 1.1, which is expected to contain the point of peak power. When a steady turbine and lorry speed is achieved, all the measurements are recorded. Each record is for 20 seconds. In a typical run, 2 to 3 such records can usually be made. For a particular brake setting, a northward and a southward run is made.

4.5.8 TESTS WITH BLUFF BODY ONLY

Driving tests were also performed with the bluff body only. However, the results were not utilised. Figure 37 shows the vehicle configured for the bluff body test. The procedure was to measure the wind velocity vector in an array of points at the lateral mid-plane above the bluff body at speeds ranging from 6 to 12 m/s.

Figure 37. Vehicle with bluff body only.

4.5.9 DRIVING TEST RESULTS AND DISCUSSION

Table 3. Uncertainties in experimental measurements.

No.	Parameter	Measurement method	Accuracy	Fractional uncertainty
1.	Brake arm length, L	CNC machine readout	± 0.03 mm	1.67×10^{-4}
2.	Load cell force, F	Chatillon force gauge	2.5%	0.0025
3.	Air density, ρ (from properties table using air temperature)	RM Young ultrasonic anemometer	2 K	0.00681
4.	Turbine area, A	Tape rule	1 mm (length)	0.002 (area)
5.	Wind speed at anemometer, V_a	RM Young ultrasonic anemometer	$\pm(1\% + 0.05)$ m/s	0.0138 (for 12 m/s runs)
6.	Turbine radius, R	Tape rule	1 mm	0.002
7.	Turbine speed, ω	Optical sensor calibrated with Tektronix oscilloscope	-	50 ppm
8.	Wheel frequency, Ω	Optical sensor calibrated with Tektronix oscilloscope	-	50 ppm
9.	Wheel circumference, c_w	Sony Xperia Arc S	53.1 m [46] (3 std. dev.)	0.0231 (2,300 m calibration distance)

The uncertainty in driving test results can be separately analysed for c_P and TSR. Using the symbols in Table 3, the expressions for c_P is:

$$c_P = \frac{\text{turbine torque} \times \omega}{\frac{1}{2}\rho A(\text{wind speed})^3}$$

$$= \frac{2 \times L \times F \times \omega}{\frac{1}{2}\rho A(\text{wind speed})^3}$$

where '2' is the sprocket ratio. For c_P based on measured wind speed,

$$c_P = \frac{2 \times L \times F \times \omega}{\frac{1}{2}\rho A V_a^3}$$

Summing the fractional uncertainties, multiplied by the index (power) where applicable, gives the overall fractional uncertainty: $1.67 \times 10^{-4} + 0.0025 + 5 \times 10^{-5} + 0.00681 + 0.002 + 3 \times 0.0138 = 0.0529$. For the cases based on vehicle speed,

$$c_P = \frac{2 \times L \times F \times \omega}{\frac{1}{2}\rho A(\Omega \times c_w)^3}$$

the fractional uncertainty is thus: $1.67 \times 10^{-4} + 0.0025 + 5 \times 10^{-5} + 0.00681 + 0.002 + 3 \times (5 \times 10^{-5} + 0.0231) = 0.0810$. These uncertainties are indicated as vertical error bars in the plots of experimental results.

As for TSR, its expression is:

$$\text{TSR} = \frac{R\omega}{\text{wind speed}}$$

and the fractional uncertainties are: $0.002 + 5 \times 10^{-5} + 0.0138 = 0.0159$ (anemometer) and $0.002 + 5 \times 10^{-5} + 0.0231 = 0.0252$ (vehicle speed). These uncertainties are indicated as horizontal error bars in the plots of experimental results.

Figure 38 shows the results for the driving tests at 12 m/s vehicle speed. An important observation was that if the vehicle speed is used as the basis for calculating c_P (shown by the blue triangles and diamonds), the plotted points scatter wildly without showing the expected trend of a quadratic curve with a maximum. This wild scatter is present even if the northbound and southbound data are considered as separate groups. Upon closer examination of the experimental data in Table 4, it can be seen that the measured wind speed at the anemometer position varies significantly even though the vehicle speed was maintained to within 0.1 m/s. This shows that existing wind and other external disturbances such as other vehicles driving past have disturbed the free stream wind speed experienced by the driving test rig. Although the existing wind was measured at the start and end of each run, it is not representative of the conditions during the test, since each test run covers a distance of about 2 km and lasts several minutes. To compensate for the effects of the free stream wind being different from the vehicle speed, the last two columns of Table 4 list the TSR and c_P calculated using the measured wind speed as the basis. When plotted, it appears as the circled data points in Figure 38, which more closely follow a quadratic curve. Similar treatment is given to the results at 10 m/s and 8 m/s and shown in Table 5 and Table 6 and Figure 39 and Figure 40. The Reynolds numbers in the table captions are based on turbine diameter of 1.1 m, air properties at 21°C and the respective nominal vehicle speeds.

Table 4. Driving test results for 12 m/s vehicle speed. $Re = 8.68 \times 10^5$.

No.	Direction	Turbine RPM	Turbine Torque (Nm)	Turbine Power (W)	Veh. Speed (m/s)	TSR (Veh)	c_p (Veh)	Measured Wind Speed (m/s)	TSR (Meas. Wind)	c_p (Meas. Wind)
1	N	136	13.72	196	12.10	0.65	0.169	12.65	0.62	0.148
2	N	160	11.76	197	12.03	0.77	0.173	12.74	0.72	0.146
3	N	158	10.69	176	12.05	0.75	0.154	12.13	0.75	0.151
4	N	236	6.33	157	12.06	1.13	0.136	13.13	1.04	0.106
5	N	249	6.95	181	12.01	1.19	0.160	13.71	1.05	0.108
6	S	185	10.13	197	12.08	0.88	0.170	12.87	0.83	0.141
7	S	172	8.16	147	12.08	0.82	0.127	11.63	0.85	0.143
8	S	195	9.96	203	12.01	0.93	0.179	12.96	0.87	0.143
9	S	195	7.55	154	12.08	0.93	0.134	11.98	0.94	0.137
10	S	224	4.91	115	12.08	1.07	0.100	12.34	1.05	0.094

Figure 38. Driving test result, 12 m/s

Table 5. Driving test results for 10 m/s vehicle speed. $Re = 7.24 \times 10^5$.

No.	Direction	Turbine RPM	Turbine Torque (Nm)	Turbine Power (W)	Veh. Speed (m/s)	TSR (Veh)	c_P (Veh)	Measured Wind Speed (m/s)	TSR (Meas. Wind)	c_P (Meas. Wind)
1	N	106	8.21	91	9.99	0.61	0.139	10.63	0.57	0.116
2	N	118	9.74	121	10.04	0.68	0.182	11.41	0.60	0.124
3	N	182	5.83	111	10.04	1.05	0.168	11.54	0.91	0.111
4	N	203	4.56	97	10.02	1.17	0.147	11.56	1.01	0.096
5	S	94	9.89	97	10.02	0.54	0.147	10.94	0.49	0.113
6	S	108	8.79	99	10.04	0.62	0.150	10.68	0.58	0.124
7	S	130	6.08	83	10.04	0.75	0.125	10.03	0.75	0.126
8	S	152	5.47	87	10.06	0.87	0.131	10.42	0.84	0.118

Figure 39. Driving test result, 10 m/s

Table 6. Driving test results for 8 m/s vehicle speed. $Re = 5.79 \times 10^5$.

No.	Direction	Turbine RPM	Turbine Torque (Nm)	Turbine Power (W)	Veh. Speed (m/s)	TSR (Veh)	c_p (Veh)	Measured Wind Speed (m/s)	TSR (Meas. Wind)	c_p (Meas. Wind)
1	N	66	6.48	45	8.01	0.47	0.133	8.50	0.45	0.111
2	N	107	5.54	62	8.04	0.77	0.183	8.87	0.70	0.137
3	N	111	6.33	74	8.00	0.80	0.220	9.53	0.67	0.130
4	N	124	3.64	47	8.05	0.88	0.138	8.52	0.83	0.116
5	N	191	1.17	23	8.06	1.37	0.068	8.99	1.23	0.049
6	N	200	0.87	18	8.00	1.44	0.054	8.67	1.33	0.043
7	S	80	4.46	37	8.08	0.57	0.109	7.70	0.60	0.125
8	S	88	4.10	38	8.07	0.63	0.110	7.64	0.67	0.130
9	S	106	3.67	41	8.04	0.76	0.119	7.87	0.77	0.127
10	S	161	3.96	67	8.04	1.15	0.196	9.97	0.93	0.103
11	S	141	3.08	46	8.05	1.01	0.133	8.69	0.94	0.106
12	S	141	2.57	38	8.04	1.01	0.111	8.32	0.97	0.101

Figure 40. Driving test result, 8 m/s

The results show an expected trend for Savonius turbines. Peak c_p was close to the expected values from prior works. As the vehicle speed was not an accurate indicator of free stream wind speed, the expected gain in c_p using the free stream wind speed as reference cannot be elucidated from these results. The next chapter explains how the use of CFD simulation complements experimental results to obtain the gain. However, these data are very valuable for validation of CFD simulation results, which is consistent with the planned methodology.

CHAPTER 5. CFD SIMULATIONS

5.1 DETERMINING REQUIRED APPARATUS FOR WIND TUNNEL TESTS

CFD simulation was used to determine the dimensions of the forward facing step in the wind tunnel for subsequent testing of a reduced scale turbine. CAD models of a step blockage of various heights in the wind tunnel test section were created. Using CFX, steady state simulations were run for the range of step height 0.1 m to 0.5 m with the settings shown in Table 7. The result for the most suitable flow field obtained, with step height of 0.2 m and wind speed of 8 m/s, is shown in Figure 41 and Figure 42. These results were validated with experimental measurements, previously described in Section 4.3. Figure 21 from Section 4.3 is replicated here for easy reference. The general shape of the wind speed profiles are consistent with experimental measurements. However, it is noted that the CFD simulation results show a higher wind speed magnitude than experimental measurements away from the centre of the re-circulation region. A point to note is that the experimental measurements were made with a Kimo VT-200F hot wire anemometer. Although the sensing wire bead itself is very small, less, than 1 mm in

Table 7. ANSYS CFX settings for determining step height in wind tunnel.

No.	Description	Value or Setting
1.	Mesh topology	3D
2.	Initial mesh face size	0.02 m
3.	Mesh adaption criterion	Velocity
4.	Mesh adaption steps	3
5.	Initial number of cells	121 thousand
5.	Final number of cells	423 thousand
7.	Step height	0.1 to 0.5 m
8.	Free stream wind speed	8 m/s and 12 m/s
9.	Advection scheme	High resolution (blend of upwind and central difference, with limits on the computed variable)
10.	Turbulence model	$k-\epsilon$
11.	Near wall treatment	scalable wall functions
12.	Fluid	Air at 25°C
13.	Pressure velocity coupling	Rhie, Chow, Majumdar

Figure 41. Flow field in wind tunnel with a bluff body installed, showing velocity contour and vector plots. Detailed view of rectangular region is shown in Figure 42.

Figure 42. Detailed view of rectangular region in Figure 41 with final mesh after three adaption steps.

Figure 21 (replicated, without error bars). Comparison of simulation and experimental results of step blockage in wind tunnel, wind speed 8m/s, step height 0.2 m.

diameter, it is located in a tunnel-like through hole on the probe shank with a diameter of 10 mm, which may have shielded it from non-longitudinal wind components, thus giving a lower reading than that predicted by CFD. At the same time, when measuring near the centre of the re-circulating bubble, the probe shank itself above and below the sensing bead may have disturbed the airflow, thus giving a higher reading than the near-zero predicted by CFD. However, as this part of the CFD simulation work was to complement experimental work whose purpose was to create a flow field over a step for subsequent testing of a reduced scale turbine, it was felt that the level of agreement between CFD and experiment as shown in Figure 21 was sufficient.

5.2 DETERMINING OPTIMUM TURBINE POSITION FOR DRIVING TEST RIG

Before the conduct of the experiment, as a supplement to the wind tunnel experiments, CFD simulation was used to preliminarily determine the optimum installation position of a turbine above a bluff body. This position can then be implemented with appropriate scaling on the driving test rig. For this purpose, a "full

Figure 43. Simulation domain and boundary conditions for determining the optimum installation position of a turbine.

size" turbine (the size proposed to the funding agency) of diameter 2.2 m and half axial length 1 m was used. The bluff body was a 5 m high two-dimensional step, which again is similar to the operating condition that the funding agency is expecting. However, the turbine is of finite width and thus the domain is considered three-dimensional. Figure 43 and Figure 44 shows the domain and mesh respectively while Table 8 shows the simulation settings, which were mostly the default settings.

Figure 44. Detailed view of the mesh for determining the optimum installation position, element size labelled.

Table 8. ANSYS CFX settings for preliminarily obtaining optimum position.

No.	Description	Value or Setting
1.	Mesh topology	3D, symmetry about turbine mid-length
2.	Turbine surfaces cell face size	0.05 m
3.	Rotating region to surroundings interface size	0.05 m
4.	Maximum cell face size	1 m
5.	Growth rate	1.1
6.	Number of cells	2.7 million
7.	Advection scheme	Upwind
8.	Transient formulation	1st order backward Euler
9.	Turbulence model	$k-\omega$
10.	Near wall treatment	Automatic switch between scalable wall functions & low-Re formulation
11.	Fluid	Air at 25°C
12.	Pressure velocity coupling	Rhie, Chow, Majumdar

Figure 45. A typical torque result from the simulation to determine optimum installation position of turbine on test rig.

The procedure was to use the parametric study feature in ANSYS Workbench to run the simulation for various horizontal (x) and vertical (y) positions of the turbine. For each position a range of TSRs from 0.6 or 0.7 to 1.1 or 1.2 was simulated. Running a simulation for a particular TSR is equivalent to loading (braking) the turbine to such an extent that the rotating speed corresponding to the said TSR is maintained by the turbine. Turbine output torque is obtained by extracting the torque about the turbine axis experienced by the turbine surfaces in the CFD simulation domain. Simulation of nine revolutions was carried out to ensure that periodicity of turbine torque had been reached. The average torque was then extracted from the final revolution. A typical torque result is shown in Figure 45. The power was calculated and plotted in Figure 46 to Figure 48. In these cases, Reynolds number based on turbine diameter and free stream wind speed was 1.71×10^6 .

Figure 46. Simulation result of power of turbine above a bluff body, $y = 1.4$.

Figure 47 Simulation result of power of turbine above a bluff body, $y = 1.6$.

Figure 48. Simulation result of power of turbine above a bluff body, $y = 1.8$.

Figure 49. Velocity flow field of the position with the highest power output: $x = 1.0 \text{ m}$, $y = 1.6 \text{ m}$.

Figure 49 shows the velocity flow field for the position with the highest power output. It is consistent with intuition that the ideal position is where the advancing blade is within the highest speed region of the separation bubble and the returning blade is shielded in the 'wind shadow' of the bluff body. From these preliminary simulations, the position with the highest power was:

$$x = 1.0 \text{ m}, y = 1.6 \text{ m}$$

or, expressed in terms of the turbine diameter D ,

$$x \approx 0.5D, z \approx 0.75D$$

The turbine position on the driving test rig was adjusted accordingly. Figure 50 shows the change. The results for $y = 1.4 \text{ m}$ are also consistent with the wind tunnel results in Figure 23 (Section 4.4). Both show that if the vertical position of the turbine axis is about $0.6D$ above the top surface of the bluff body, for maximum power output, the corresponding axis horizontal position is coincident with the front face of the bluff body, i.e., $x = 0 \text{ m}$. This is shown in Figure 46.

Figure 50. Old position of turbine (left) compared to new position (right).

5.3 VALIDATION WITH WIND TUNNEL TEST OF WIND TURBINE

Saha [23] *et al.* tested a three-bladed Savonius turbine with radius of 0.135 m in an open jet wind tunnel. An extract of their results (those closest to the wind speeds in the present research) is shown in Table 9. These independent experimental results were used for the validation of simulation settings that were applied for the simulation of driving test conditions. As the turbine is a cross flow device and not an axial flow one, its wake is not symmetrical. Several trials were performed with varying asymmetry of the domain so that the final selected domain captures the entire wake. As seen in Figure 51, the velocity contour for 5% of the wind speed is well within the boundaries of the domain. The domain is a three-dimensional one that models half of the experimental apparatus, since there is a horizontal plane of symmetry at the mid-length of the turbine. Details of the settings are shown in Table 10 and the mesh settings are illustrated in Figure 52. A velocity contour plot of a typical result is shown in Figure 51. A typical y^+ plot is shown in Figure 53, showing maximum y^+ of about 100 which is well within the recommended range of 0.2 to 300 for the SST turbulence model with automatic near wall treatment.

The paper reported no-load (un-braked) turbine RPM at various wind speeds. To estimate the frictional torques of the experiment, simulation of these no-load speeds were performed. The torques obtained by simulation are thus the frictional torques of the experimental apparatus. The frictional torques obtained were used to adjust the ideal simulation results to obtain the experimental performance curve which showed a similar maximum c_p with the experiment, as shown in Figure 54. This validation confirms the suitability of the simulation settings used.

Table 9. Extract of results of wind tunnel tests on three-bladed Savonius turbine by Saha et al.

No.	Wind Speed (m/s)	Re (based on turbine $\varnothing$)	Maximum c_p	(for estimating frictional torque)	
				No-load RPM	No-load TSR
1.	10.17	1.78×10^5	0.0822	480	0.667
2.	9.48	1.66×10^5	0.0893	461	0.696
3.	8.23	1.44×10^5	0.1104	439	0.754

Table 10. ANSYS CFX settings for simulating the experiment of Saha et al.

No.	Description	Value or Setting
1.	Mesh topology	3D, symmetry about turbine mid-length
2.	Domain dimensions (L×W×H)	1.35 m × 1.755 m × 0.685 m
3.	Wake region dimensions (L×W×H)	0.875 m × 0.335 m × 0.17 m
4.	Turbine surfaces cell face size	4 mm
5.	Rotating region to wake region interface size	7 mm
6.	Wake region cell body size	7 mm
7.	Maximum cell face size	0.11552 m
8.	Growth rate	1.1
9.	Number of cells	3.09 million
10.	Advection scheme	High resolution (blend of upwind and central difference, with limits on the computed variable)
11.	Transient formulation	Second order backward Euler
12.	Time step	1/120 of rotational period
13.	Turbulence model	Shear stress transport
14.	Near wall treatment	Automatic switch between scalable wall functions & low-Re formulation
15.	Fluid	Air at 25°C
16.	Pressure velocity coupling	Rhie, Chow, Majumdar

Figure 51. CFD simulation of experiments by Saha et al., typical velocity contour plot.

Figure 52. CFD simulation of experiments by Saha et al., bottom view of domain mesh (partial detailed view).

Figure 53. CFD simulation of experiments by Saha et al., typical y^+ plot, wind speed 9.48 m/s, TSR 0.5.

Figure 54. CFD simulation of experiments by Saha et al.

5.4 SIMULATION OF DRIVING TEST CONDITIONS

The driving test results were used for validation of CFD simulation. Accordingly, simulation of the conditions of the driving test using ANSYS 13 Workbench and CFX was set up and run. Table 11 lists the settings. The geometry was constructed in SolidWorks. It is a three-dimensional domain that models half of the experimental apparatus, since there is a vertical plane of symmetry and consists of stationary domains for the surrounding air and wake region and a rotating domain for the region around the turbine. Solid bodies such as the turbine and lorry are represented by voids in those domains whose surfaces are defined with wall boundary conditions. Figure 55 shows the velocity contour plot of a typical simulation result for free stream wind speed (also vehicle speed) of 12.08 m/s and TSR of 0.883. Compared to Figure 49, the separation bubble has a lower maximum speed due to the finite width of the test rig bluff body. For the same reason, the wind speed is restored to near free stream value after the turbine.

Table 11. ANSYS CFX settings for simulating the driving test conditions.

No.	Description	Value or Setting
1.	Mesh topology	3D, symmetry about turbine mid-length
2.	Domain dimensions (L×W×H)	15 m × 9 m × 10 m
3.	Wake region dimensions (L×W×H)	6.8 m × 2.4 m × 1.5 m
4.	Turbine surfaces cell face size	0.02 m
5.	Rotating region to wake region interface size	0.02 m
6.	Wake region cell body size	0.06 m
7.	Other surfaces cell face size	0.06 m
8.	Maximum cell face size	0.5 m
9.	Growth rate	1.1
10.	Number of cells	2.96 million
11.	Advection scheme	High resolution (blend of upwind and central difference, with limits on the computed variable)
12.	Transient formulation	Second order backward Euler
13.	Time step	1/120 of rotational period
14.	Turbulence model	Shear stress transport
15.	Near wall treatment	Automatic switch between scalable wall functions & low-Re formulation
16.	Fluid	Air at 25°C
17.	Pressure velocity coupling	Rhie, Chow, Majumdar

Figure 55. Typical CFD simulation result showing the velocity contour plot. The blue octahedron indicates the position of the anemometer.

5.4.1 DOMAIN SIZE AND GRID INDEPENDENCE STUDIES

The final domain size and grid fineness shown in Table 11 were obtained after the following independence studies. First of all, the appropriate domain size was determined. Having a large domain would simulate the actual driving test condition more closely. However, it is impractical due to computational costs. So the domain is progressively enlarged until the simulation result does not change, meaning that the result is as good as using any larger domain, including one that is as large as the actual driving test condition. Figure 56 shows the variation in wind load and torque results obtained from CFD simulation of driving test conditions, using different domain sizes. Observation was also made on the velocity contours. Each contour band is 0.25 m/s. One of the criteria for selecting domain size was that the wind speed at the domain boundaries does not deviate more than 5% from the free stream wind speed. From the results, a domain size of 9 x 10 m was chosen. The domain was also chamfered at its upper-outer corner as it was noted that the wind speed bands were circular and

reducing the domain at the upper outer corner does not cause the wind speed at that boundary to exceed the deviations at the top and side.

From the earlier simulation of a forward facing step in the wind tunnel (see section 5.1), whose result was validated by actual wind tunnel measurements (see section 4.3.2), it was noted that in the re-circulating region ahead of the step, the point at which the wind velocity vector begins to deviate from horizontal is less than two step-heights ahead of the step. Hence the domain space ahead of the turbine axis was set as 5 m. From prior literature, the re-attachment point of the re-circulation bubble above a forward facing step is less than five times the step-height. Hence the domain space after the test rig frontal surface was set as 10 m.

Figure 56. Simulation of driving test conditions, variation of wind load and torque with domain size.

With the domain size found, a grid independence study was performed. The edge length of the elements adjacent to the turbine surfaces is set to 0.02 m (2% of turbine diameter). The turbine is set in a rotating region. The interface between the rotating region and the surrounding wake region also has elements with edge length of 0.02 m.

In the wake region, element edge length is allowed to grow at a rate of 1.1 to a variety of maximum sizes ranging from 0.0355 m to 0.15 m. The size of the wake region is 6.8 m $\times$ 2.4 m $\times$ 1.5 m. Lengthwise, it is approximately 6 times the turbine diameter while height of the region above the top of the turbine is approximately equal to the turbine diameter. Width-wise, the region extends by a distance equal to the turbine axial length from the end of the turbine. Beyond the wake region, element edge length at the interface with the surrounding air is set to the same dimension as the wake region elements. From this interface, the element edge length is allowed to grow, also at a rate of 1.1 to a maximum size of 0.5 m in the surrounding air. All other solid surfaces such as the bluff body surface have element length of 0.06 m, which was selected to be 50% larger than the cross-sectional length of the aluminium extruded profiles that the test rig is constructed with. Figure 57 shows the three regions that form the whole simulation domain.

Figure 57. Simulation of driving test conditions, the three regions that form the simulation domain.

Figure 58. Simulation of driving test conditions, overview of mesh (above) and detail of turbine region (below).

Having a fine mesh or grid would better capture fluctuations of flow properties, including the spatially finer ones, thereby giving a result more similar to reality. However, an excessively fine grid is computationally costly. Hence, the grid fineness is varied by varying the element edge length of the wake region. Beyond a certain level of fineness, the result does not change, meaning that the result is as good as any finer grid. The result of this grid independence study is shown in Figure 59. From the results, wake region element edge length of 0.06 m, or approximately 6% of turbine diameter, was

Figure 59. Simulation of driving test conditions, variation of wind load and torque with grid fineness.

Figure 60. Simulation of driving test conditions, typical y^+ , wind speed 12.081 m/s, TSR 0.88306.

chosen. Throughout this research, meshing was performed automatically by ANSYS meshing which ensures that the mesh is of adequate quality. As an example, in Figure 58 the majority of the elements, over 96%, have an orthogonal quality better than 0.75.

Figure 60 is a plot of the wall y^+ for a typical simulation case, showing values slightly above 300. The Automatic Near Wall Treatment implemented with the SST turbulence model is able to cope with y^+ ranging from 0.2 (where the low-Re formulation is applied) to 300 (where the wall function is applied). Ultimately, the average torque output predicted by CFD agrees well with experimental results (see next section) and thus reliance on CFD can be made with confidence.

5.4.2 COMPARISON WITH OTHER TURBULENCE MODELS

Although the Shear Stress Transport $k-\omega$ turbulence model had been selected for performing simulations after a review of its recommended application in Section 2.5, two other turbulence models, the widely used $k-\epsilon$ and the Spalart-Allmaras models were also used in simulations of a typical driving test condition. Figure 61 shows the result. The torques in Table 12 are obtained by taking the average of the respective torque plots. The SST model gives a half-turbine torque (magnitude) of 4.06 Nm while the $k-\epsilon$ model under-predicts the torque quite significantly, giving 3.67 Nm. The Spalart-Allmaras model gives 3.92 Nm, quite close to the SST result. However, the solution time takes slightly longer, in spite of being a one-equation model. For reference, the experimental result for the same TSR of 0.98 (based on wind speed at anemometer of 13.14 m/s) is 4.08 Nm, from the second-order polynomial curve fitted to Figure 38.

Table 12. Comparison of three turbulence models with experimental result.

No.	Turbulence Model	Torque (Nm)
1.	SST	4.06
2.	$k-\epsilon$	3.67
3.	Spalart-Allmaras	3.92
4.	- Experimental result	4.08

Figure 61. Simulation of a typical driving test condition using three different turbulence models.

5.4.3 VALIDATION WITH DRIVING TEST RESULTS

With the above simulation setup, CFD simulation of the driving test conditions was carried out. The simulated wind speeds and TSRs were taken from the table of experimental results. Table 13 to Table 15 list the CFD results for free stream wind speed of 12, 10 and 8 m/s respectively. As previously explained, the vehicle speed is not an accurate indication of the free stream wind speed. Hence the means of comparing and validating the CFD results with the experimental results is by using the wind speed at the anemometer position as a common reference. The columns shaded in pink in Table 13 to Table 15 show the wind speed obtained by CFD at the anemometer position and the TSR and c_p based on this speed. The latter two parameters are plotted together with the experimental results in Figure 62 to Figure 64.

Table 13. CFD simulation result for 12 m/s vehicle speed (also free stream wind speed).

No.	TSR (User-Selected)	Turbine RPM	Turbine Torque After Subtracting Friction (N)	Turbine Power (W)	Free Stream Wind Speed (m/s)	c_P	CFD Wind Speed @ Anemometer (m/s)	TSR (Based on Wind Speed)	c_P (Based on Wind Speed)
1.	0.64958	136	14.36	205	12.100	0.177	13.183	0.60	0.137
2.	0.75319	158	12.88	212	12.046	0.186	13.112	0.69	0.144
3.	1.1947	249	6.11	159	12.011	0.141	13.069	1.10	0.109
4.	0.88306	185	10.63	206	12.081	0.179	13.146	0.81	0.139
5.	0.93403	195	9.90	202	12.010	0.178	13.066	0.86	0.139
6.	1.0695	224	7.90	186	12.082	0.161	13.143	0.98	0.125

Figure 62. Comparison of CFD and experimental result: 12 m/s.

It is observed that for 12 m/s vehicle speed, there is good agreement between the experimental and CFD results while at 10 and 8 m/s there was poor agreement. It is hypothesised that cross wind was the cause of lower experimental c_P . Thus a basis to identify and exclude such experimental data with cross wind has to be found. The

prevailing wind was measured at occasional intervals during the experiment. A plot of the lateral component of wind is shown in Figure 65. It can be seen that the magnitude of the lateral component of wind, though small, may fluctuate rapidly. Therefore it is not a good indicator of the cross wind during any particular run.

Table 14. CFD simulation result for 10 m/s vehicle speed (also free stream wind speed).

No.	TSR (User-Selected)	Turbine RPM	Turbine Torque After Subtracting Friction (N)	Turbine Power (W)	Free Stream Wind Speed (m/s)	c_p	CFD Wind Speed @ Anemometer (m/s)	TSR (Based on Wind Speed)	c_p (Based on Wind Speed)
1.	0.61033	106	10.27	114	9.9942	0.174	10.89	0.56	0.135
2.	1.1680	203	4.57	97	10.025	0.148	10.91	1.07	0.114
3.	0.53811	94	11.48	113	10.024	0.171	10.94	0.49	0.131
4.	0.74722	130	8.99	123	10.039	0.185	10.93	0.69	0.144
5.	0.87261	152	7.46	119	10.060	0.179	10.95	0.80	0.139

Figure 63. Comparison of CFD and experimental result: 10 m/s.

Table 15. CFD simulation result for 8 m/s vehicle speed (also free stream wind speed).

No.	TSR (User-Selected)	Turbine RPM	Turbine Torque After Subtracting Friction (N)	Turbine Power (W)	Free Stream Wind Speed (m/s)	C_P	CFD Wind Speed @ Anemometer (m/s)	TSR (Based on Wind Speed)	C_P (Based on Wind Speed)
1.	0.47408	66	7.71	53	8.0119	0.158	8.76	0.43	0.121
2.	0.7697	107	5.52	62	8.0441	0.183	8.76	0.71	0.142
3.	0.88353	124	4.66	60	8.0515	0.176	8.76	0.81	0.137
4.	1.3677	191	1.42	29	8.0606	0.083	8.76	1.26	0.065
5.	0.63066	88	6.43	60	8.0678	0.173	8.79	0.58	0.134
6.	1.1539	161	2.95	50	8.039	0.146	8.75	1.06	0.114

Figure 64. Comparison of CFD and experimental result: 8 m/s.

Upon closer examination of the experimental results, it is found the presence or absence of significant cross wind can be obtained by comparison of the anemometer readings of two consecutive runs in opposite directions. Referring to Figure 66 which is a sketch of the plan view of the test rig with the position of the anemometer indicated, it is expected

Figure 65. Lateral component of prevailing wind.

Figure 66. Sketch of plan view of test rig: no cross wind (top); with cross wind, driving left to right (middle); with cross wind, driving right to left (bottom).

that the oncoming wind, after striking the frontal face of the test rig, will flow not directly rearward, but slightly outward, deviating by a small angle away from the mid-plane of the lorry. This angular deviation from mid-plane is presented in the column “Wind direction” in Table 16 to Table 18. If there is no cross wind, then this angle should be the same for a southward and a northward run. A difference in the angle in two consecutive runs indicates otherwise. The 12 m/s runs were conducted from 4:05 am to 5:17 am on 22 Oct 2012. Referring to Table 4 in Chapter 4, for typical consecutive northward and southward runs, the lateral angle of the wind at the anemometer position is compared. For example, row 3 results were recorded at 4:36 am and row 6 results were recorded at 4:39 am. The lateral direction of the measured wind was 7.4 and 7.3 degrees respectively, with zero degrees being the straight rearward direction. As shown in Figure 62, these two data points fall closely on the CFD result curve. Figure 62 has two points with poor agreement with CFD results, the points for row 4 and row 10 of Table 4. Examining their lateral wind directions gave 8.5 and 5.0 degrees respectively. The remainder of the lateral wind directions of the 12 m/s runs are listed in Table 16. The red row indicates large cross wind while the green row indicates little cross wind.

Table 16. Comparison of lateral wind direction of consecutive runs, 12 m/s.

Row No. of Table 4	Run No.	Time	Run direction	Wind direction (deg)	Row No. of Table 4	Run No.	Time	Run direction	Wind direction (deg)
4	27	04:07	N	8.46	10	25	04:04	S	4.95
5	29	04:16	N	7.59	9	28	04:14	S	8.43
5	29	04:16	N	7.59	7	31	04:21	S	7.25
3	33	04:36	N	7.41	8	32	04:29	S	8.24
3	33	04:36	N	7.41	6	34	04:39	S	7.28
2	35	04:42	N	8.37	6	34	04:39	S	7.28
1	36	05:17	N	6.09					

In comparison, for the 8 m/s runs, which were conducted earlier in the day from 2:02 am to 3:45 am, typical consecutive runs of opposite direction in Table 6 row 3, recorded at 2:25 am, and row 8, recorded at 2:28 am, gave lateral direction of the measured wind of 6.4 and 7.8 degrees respectively. The 8 m/s case implies a greater cross wind during the

experimental period because the lateral direction of the measured wind in opposing runs had a greater difference. Similarly, for the 10 m/s runs which were conducted on 19 Oct2012 from 3:33 am to 4:11 am, consecutive runs in opposite directions in Table 5 row 1, recorded at 3:33 am, and row 6 , recorded at 3:36 am, gave lateral direction of the measured wind of 7.5 and 8.8 degrees respectively, again implying a greater cross wind. At these two wind speeds, most of the runs had significant cross wind and hence there was poor agreement with CFD results. The comparison of lateral wind direction for these two runs are in Table 17 and Table 18. With the above explanation, the author is satisfied that the experimental results for cases where cross wind is not significant can be used to validate the CFD results.

Table 17. Comparison of lateral wind direction of consecutive runs, 10 m/s.

Row No. of Table 5	Run No.	Time	Run direction	Wind direction (deg)	Row No. of Table 5	Run No.	Time	Run direction	Wind direction (deg)
1	22	03:33	N	7.50	6	23	03:36	S	8.85
2	25	03:53	N	6.05	5	24	03:46	S	4.98
3	27	04:00	N	9.37	7	26	03:57	S	7.00
4	29	04:11	N	8.46	8	28	04:08	S	4.16

Table 18. Comparison of lateral wind direction of consecutive runs, 8 m/s.

Row No. of Table 6	Run No.	Time	Run direction	Wind direction (deg)	Row No. of Table 6	Run No.	Time	Run direction	Wind direction (deg)
6	4	02:02	N	7.47					
					10	5	02:16	S	6.14
3	7	02:28	N	6.39	8	6	02:25	S	7.83
2	8	02:29	N	8.32	7	9	02:36	S	6.90
1	10	02:42	N	7.95	7	9	02:36	S	6.90
1	10	02:42	N	7.95	12	11	02:48	S	5.73
9	13	02:54	N	8.20	7	12	02:49	S	5.77
5	21	03:45	N	6.75	11	20	03:40	S	4.14

5.4.4 SUMMARY OF APPLICABLE SIMULATION SETTINGS

In summary, ANSYS CFX can be used to reliably estimate the performance of a metre-sized Savonius turbine using the settings shown in Table 19. However, the settings involving the mesh sizing on the turbine surfaces have only been validated by a turbine diameter of 1.1 m. Excessively exceeding the validated size will at some point introduce unacceptable errors in the CFD results due to increased y^+ . The corresponding Reynolds number based on turbine diameter and free stream wind speed of 12 m/s is about 8.5×10^5 .

Table 19. Summary of applicable simulation settings.

No.	Description	Value or Setting
1.	Domain length	5D (turbine diameter) upstream 10D downstream
2.	Domain width (axial direction of turbine)	18(L/2) (turbine half width)
3.	Domain height (radial direction of turbine)	6D above turbine axis
4.	Wake region length	6D
5.	Wake region width	1.5D
6.	Wake region height	1D above top of rotating region
7.	Rotating region size	0.02D larger than turbine all round
8.	Turbine surfaces cell face size	0.02D
9.	Rotating region to wake region interface size	0.02D
10.	Wake region cell body size	0.06D
11.	Maximum cell face size	0.5D
12.	Growth rate	1.1
13.	Number of cells	> 3 million
14.	Transient formulation	Second order backward Euler
16.	Time step	1/120 of rotational period
15.	Advection scheme	High resolution (blend of upwind and central difference, with limits on the computed variable)
17.	Turbulence model	Shear stress transport
18.	Near wall treatment	Automatic switch between scalable wall functions & low-Re formulation
19.	Fluid	Air at 25°C
20.	Pressure velocity coupling	Rhie, Chow, Majumdar

5.5 OPTIMISATION OF A GENERIC TURBINE

A comprehensive computational fluid dynamics simulation study was carried out to determine the optimum position of the turbine with step height, turbine position and TSR as the parameters to be varied. The details are explained in the following sections.

5.5.1 CHOICE OF STEP HEIGHT AND DOMAIN SIZE

The purpose of this part of work is to determine the range of heights of bluff body that should be simulated, and for each of those heights, what domain size to use. As it is expected that the knowledge gained from this work will be used to determine the optimum installation position of a Savonius turbine on the roof of a cuboidal building, the choice of step height to be studied has to be useful for this practical application.

Table 20. ANSYS CFX settings for simulating bluff buildings of various heights, equivalent to a two-dimensional forward facing step.

No.	Description	Value or Setting
1.	Mesh topology	2D
2.	Domain dimensions (L×H)	175 m × 100 m
3.	Wall element size	0.05 m
4.	Maximum face size	2.0 m
5.	Growth rate	1.1
6.	Initial number of cells	88,091
7.	Mesh adaption criteria	Pressure, velocity
8.	Adaption steps	3
9.	Method of refinement	Subdivide into 2
10.	Adaption method	Solution variation
11.	Max. iteration per adaption step	300
12.	Final number of cells	247,397
13.	Advection scheme	High resolution (blend of upwind and central difference, with limits on the computed variable)
14.	Turbulence model	Shear stress transport
15.	Near wall treatment	Automatic switch between scalable wall functions & low-Re formulation
16.	Fluid	Air at 25°C
17.	Pressure velocity coupling	Rhie, Chow, Majumdar

Considering that the height of one floor of a building is typically 2.8 m, and that two-storey buildings are the typical minimum height, a roof height of about 6 m is likely to be the minimum height that is encountered in practice. Thus the choice of 5 m, 10 m and 15

m was selected for study to encompass the expected minimum roof height. Interpolation or extrapolation of behaviour can then be made for other heights beyond the range that was studied.

Using steady state simulation of a two-dimensional forward facing step, with the simulation settings given by Table 20 and geometry of the domain generally illustrated by Figure 67, the velocity flow field was obtained. The domain was progressively enlarged, as shown in Figure 68 to Figure 70, until further enlargement does not produce significant change in the separation bubble ahead and behind the step, defined by a shift of the zero wind speed line of less than 10% of the bubble length. In addition, a symmetry boundary condition was applied at the top of the computational domain, and the domain size has to be sufficiently large so that no inflow condition appears at the boundary. Figure 67 also shows the mesh of the chosen domain dimensions after three mesh adaption (refinement) steps and the boundary conditions.

Figure 67. Simulation to support choice of step height: mesh after refinement and boundary conditions.

Figure 68. Variation of domain length after the step, from left to right: 4x, 12x, and 20x step height. Note complete inclusion of the separation bubble at 20x.

Figure 69. Variation of domain total height, from left to right: 10x, 20x, and 40x step height. Note insignificant difference between 20x and 40x.

Figure 70. Variation of domain length before the step, from left to right: 6x, 9x, and 15x step height. Note disappearance of inflow for 15x case.

The conclusion drawn from these simulations is given in Table 21. Using these dimensions, simulations were repeated for step height of 10 m and 15 m. Contour plots of the flow fields are given in Figure 71. Simulations using domains that were 20% larger was also carried out to ensure that the flow field indeed does not vary significantly from what is deemed suitable, as shown in Figure 72.

Table 21. Suitable domain size for simulation of flow over a forward facing step.

Description	Value or Setting
Domain length after step	20 times step height
Domain length before step	15 times step height
Total domain height	20 times step height

Figure 71. Contour plots above a forward facing step at 12 m/s free stream wind speed for step height of 10 m (left) 15 m (right)

The wind speed profiles of the three step heights were extracted and plotted in Figure 73, Figure 74 and Figure 75. Note that the plots are very similar when normalised using the step height. It shall be later explained that the line of minimum speed is of most interest for expressing the line of optimum turbine positions.

Figure 72. Simulation to support choice of step height: comparison of velocity profiles for suitable size and 20% larger size.

Figure 73. Wind speed profile for step height of 5 m, free stream wind speed of 12 m/s.

Figure 74. Wind speed profile for step height of 10 m, free stream wind speed of 12 m/s.

Figure 75. Wind speed profile for step height of 15 m, free stream wind speed of 12 m/s.

5.5.2 VALIDATION WITH PRIOR LITERATURE

The settings used for the preceding step height simulations, which includes three steps of mesh adaption (refinement) and the use of the Shear Stress Transport turbulence model, were validated by simulating the experiment performed by Baker [42]. He performed wind tunnel tests of a two-dimensional forward-facing step and measured wind velocity using thermal anemometry. The wind tunnel test section had a height of 1.07 m and the step had a splitter board upstream and height of 0.076 m. The geometry of the setup is shown in Figure 76. One of the findings from his results was that the re-attachment of the separation bubble above the step occurred at $4.8H$ behind the edge of the step. He used both the hot-wire anemometer and the pulsed wire anemometer. However, he concedes that hot-wire anemometry is unsuitable for measuring the highly turbulent reversed flow region (i.e., within the separation bubble) and that he had no means to independently verify the validity of the pulsed wire anemometer measurements in that region. His result for the longitudinal component of wind velocity is used for comparison with CFD simulation, shown in Figure 78. It is noted that good agreement between the simulation results and Baker's result is seen, with the peak velocity and regions further from the separation bubble almost identical. The re-attach-

Figure 76. CFD simulation of experiment by Baker, longitudinal velocity component plot. The lines are measurement locations.

Figure 77. CFD simulation of experiment by Baker, showing the mesh after three adaption steps and boundary conditions.

Figure 78. Longitudinal velocity from CFD simulation of Baker's work compared with his experimental results.

ment point is also identical, at five step heights from the edge. What is different is that the CFD simulation predicts a smaller bubble height of about 20% lower in the vertical

direction compared to experiment. Indeed, if the CFD data were stretched in the vertical direction by a corresponding amount, almost perfect agreement would be obtained. It appears that the CFD code produces an artificially high viscosity of the fluid, thus resulting in steeper velocity gradients than reality.

Figure 79. Normalised turbulent kinetic energy from CFD simulation of Baker's work compared with his experimental results.

Figure 79 shows the normalised turbulent kinetic energy of the flow above the step in Baker's experiment. As previously discussed in the literature review chapter, Section 2.5, the Shear Stress Transport (SST) turbulence model employed uses a limiter on the rate of production of turbulent kinetic energy. Hence no over prediction of turbulent kinetic energy is observed. Rather, under prediction by nearly 50% after re-attachment is observed. An observation similar to that for the longitudinal velocity profile plot is that the peaks in the turbulent kinetic energy are also about 20% lower in the vertical direction than experimental peaks, thus ensuring that the pattern of turbulence levels within the separation bubble, other than appearing compressed vertically, is generally consistent with experimental observations. It should be noted that for the present research involving a Savonius turbine, the mean flow and the transfer of air momentum

to the turbine to produce power is more important and hence these errors in the turbulence levels are acceptable for the scope of this research.

On the whole, the general agreement between CFD and experiment of the longitudinal velocity and to a lesser extent, turbulent kinetic energy confirms the reliability of the simulation settings used for determining the flow field above a forward facing step.

5.5.3 PARAMETRIC SIMULATION RESULTS

Combining the requirements listed in Table 19 and Table 21, parametric simulations of a generic turbine above a step was performed. The mesh settings and wake region size are set according to Table 19 while the overall domain size is set according to Table 21. A turbine diameter of 2.5 m was chosen as it is typical of the largest commercially available Savonius turbines [14], [30]. For a typical case with the turbine at a horizontal position of $x = 2R$ and vertical position $y = 2.5$ m, $TSR = 1.3$ and step height of 5 m, the domain is shown in Figure 80, which is a three dimensional domain with a vertical plane of symmetry at the turbine mid-length. The half-length of the turbine is 2.5 m because it offers the optimum efficiency according to Menet [45].

Figure 80. Domain for parametric simulation of generic turbine above a bluff body (forward facing step).

Figure 81. Overview of mesh, generally showing the underside of the domain looking from the outlet end.

Figure 82. Mesh of wake and rotating region.

A frozen rotor simulation is run until convergence of the steady state solution is achieved to obtain the flow field within a relatively short computational time. The result of this simulation is used as the initial condition for a transient simulation. The torque and wind load produced by the transient simulation is then extracted. By trial and error, it was found that simulating for 2.5 revolutions generally gives a torque result that is

Figure 83. Typical torque and wind load result. Dotted lines show the average values over one revolution.

Figure 84. Velocity contour plot of the turbine and wake region. View angle: 15° azimuth from turbine axis, 15° elevation.

Figure 85. Pressure contour plot of the turbine and wake region. View angle: 15° azimuth from turbine axis, 15° elevation.

periodic. For some cases, especially those where the turbine position is near the top surface of the bluff body, simulating more than 2.5 revolutions was necessary to observe periodic torque. In Figure 83, the dotted lines are the moving average of the preceding one revolution of data points. In this typical case, the moving average at the 2.5 revolution mark is within about 10% of the average torque for the final period which is taken as the final steady average torque. The average torque calculated using data points from 1.5 to 2.5 revolutions is used for calculation of power and its coefficient. A typical velocity and pressure contour plot is shown in Figure 84 and Figure 85.

To examine the effect of wall y^+ on the results, two different meshes were prepared. The left side of Figure 86 shows the mesh without inflation layers. The cell face size at the turbine surfaces was 0.11 m due to global limiting, while the right side shows the mesh with 5 inflation layers, the thickness of each layer being 0.1 of the face size of 0.05 m. For the former case, the wall y^+ was as high as 1800, while in the latter case, most areas were below 300, as shown in Figure 87. However, the torque results of both cases do not differ significantly, as shown by the plots of torque and wind load in Figure 83, with the average torque at the 2.5 revolution mark differing by only 3.5% (318 Nm vs. 307 Nm) in spite of the additional computational time of 51% (25 hours vs. 16.5 hours)

for the inflated case. Thus it is felt that omission of inflation layers in the remaining cases is justified.

Figure 86. Mesh near turbine surfaces, without inflation (left), with inflation (right).

Figure 87. Wall y^+ without inflation layers (left), with inflation layers (right).

The process is repeated for a range of turbine axis positions as shown in the sketch in Figure 88. For each position, a range of TSR is simulated. The TSR range is chosen to ensure that the two-peak phenomena (where present) is elucidated or that the c_p decreases monotonically at two ends of the range. The lower limit of the TSR range is 0.6 to 0.8, while the upper limit is 1.2 to 1.6, depending on the position of the turbine. The

full results for step height of 5 m and turbine diameter of 2.5 m is shown in Figure 89. At each position, there is a peak c_P . Referring to the sketch of positions in Figure 88, each column at the same horizontal position would give three peak c_P values. These peak c_P values are plotted against vertical position and thus the optimal vertical position can be obtained by finding the maximum of the fitted quadratic curve. Repeating the plots for all four horizontal positions give the line of peak c_P , as shown in Figure 90. Setting the criterion for optimal installation position as that which maximise c_P , this line is also the line of optimal installation position. Comparison with the line of minimum velocity of the flow field over the bluff body will be made in the next chapter.

Figure 88. Sketch of the turbine positions that were simulated. R is turbine radius ($= 1.25$ m).

Figure 89. c_p vs TSR for a range of turbine positions, for turbine diameter of 2.5 m, bluff body step height of 5 m and free stream wind speed of 12 m/s

Figure 90. Line of peak c_p , also the line of optimal installation position.

1.1.1 SUMMARY OF GENERALISED CFD METHODOLOGY

The above descriptions of simulation set up can be used as the methodology for finding the line of optimal installation position for other values of parameters such as turbine diameter, wind speed and step height. In summary, for a given turbine size, the settings in Table 19 should be used, with the exception of domain size, which should depend on step height according to Table 21. For each position to be simulated, the TSR range should be chosen such that the c_p decreases monotonically at both limits of the range. The simulation should start with a frozen rotor analysis to obtain the initial flow field for a subsequent transient analysis. The simulated duration should be at least 2.5 revolutions or sufficient length such that periodicity of torque is achieved.

CHAPTER 6. EMPIRICAL MODEL

6.1 DEVELOPMENT OF THE EMPIRICAL RELATIONSHIP

Examination and comparison of the various CFD simulation results leads to the development of an empirical relationship between the line of the optimal turbine axis position and the line of minimum wind speed in the flow field over a bluff body.

6.1.1 OVERVIEW OF THE EMPIRICAL RELATIONSHIP

The plot of wind speed lines in section 5.5.1 and the line of peak c_p in section 5.5.3 is converted into the same basis (i.e., multiples of bluff body step height) and superimposed. The outcome is shown in Figure 91 below.

Figure 91. Line of peak c_p superimposed on wind speed lines, $H = 5$ m, $V_\infty = 12$ m/s, $R = 1.25$ m.

It is immediately apparent that the shape of the line of peak c_p is similar to the line of minimum wind speed, with a vertical offset of about $+1.1R$, where R is the turbine radius. Table 22 lists the positions of peak coefficient of power, c_p . The c_p value begins at 0.573 at the front most position $0.25H$ from the front edge of the bluff body, rising to

0.601 at $0.50H$, and subsequently decreasing slightly to 0.526 at $1.00H$. Closer examination of the superimposed velocity contour diagram in Figure 91 shows that at $0.25H$, the maximum wind speed is not as high as at other positions, thus explaining the lower c_p . Beyond $0.5H$, the optimum position for the turbine axis begins to dip below the line of maximum wind speed. At the same time, c_p decreases. This seems to contradict the intuitive expectation that a high torque and hence high c_p would be obtained if the advancing blade (moving with the wind) of the turbine is located somewhere near the region of maximum wind speed, i.e., the turbine axis is somewhere below the line of maximum wind speed. To explain this apparent counter-intuition, a closer examination of the torque at various turbine angular positions is made in the next few sections.

Table 22. Positions of peak coefficient of power

No.	Horizontal position from front edge of bluff body in terms of:		Vertical position above bluff body in terms of:		c_p
	bluff body height H	turbine radius R	bluff body height H	turbine radius R	
1.	0.25	1	0.443	1.77	0.573
2.	0.50	2	0.520	2.08	0.601
3.	0.75	3	0.564	2.26	0.591
4.	1.00	4	0.624	2.49	0.526

6.1.2 THE SOURCE OF TORQUE PRODUCTION AT LOW TSR

Referring to the c_p vs. TSR plot for turbine position of $x = 1R$, $y = 2.0$ m in Figure 89 (highlighted in Figure 92), it is noted that there are two distinct local peaks at TSRs of 0.8 and 1.2. This is due to the change of relative dominance of drag-based torque at lower TSR and lift-based torque at higher TSR. This phenomenon of lift in a supposedly drag-based turbine has been explained by Zhou and Rempfer [47]. To elaborate on the case of $TSR = 0.8$, the turbine torque over one revolution is plotted in Figure 93. It is noted that the point of lowest magnitude of torque corresponds to the turbine angular position where the concave side of the advancing blade faces almost perpendicularly to the oncoming wind. Examining the pressure contour plot on the top left of Figure 93, it is seen that there is a slight stagnation of the wind in the concave side of the advancing

blade, shown in light green. On the convex side of the advancing blade, pressure is low only near the tip (shown in dark blue), caused by a separation bubble (evident in the velocity contour plot at the bottom left of Figure 93), while the major part of it is at moderately low pressure (shown in cyan). Stagnation is moderated by the blade travelling in the same direction as the wind. This produces torque in the intended direction of rotation. However, on the convex surface of the returning blade, severe stagnation occurs, giving a high pressure region shown in red. Stagnation is strengthened by the blade travelling against the wind. This high pressure, coupled with the low pressure on the concave side of the returning blade shown in cyan/light blue, gives rise to a torque opposing the intended direction of rotation. The net result is that the torque production in the intended direction of rotation is low.

Figure 92. Extract from Figure 89, highlighting two peaks in the C_p vs. TSR curve for turbine position $x = 1R$, $y = 2 m$.

The turbine angular position that gives the greatest torque production is when the concave side of the advancing blade faces slightly downward, as shown in the contour plots on the right side of Figure 93. From the pressure plot, it is seen that greater stagnation occurs inside the concave side of the advancing blade, with a slight orange colour. With the blade at this angle, the component of blade speed parallel to the wind direction is lower, thus enhancing the stagnation. On its convex side, almost the entire surface is at low pressure (shown in dark blue) due to the high speed flow over the

Figure 93. Torque over one revolution, TSR 0.8.

surface producing lift by the Bernoulli Effect. These two effects combine to give a large torque in the intended direction of rotation. The flow over the advancing blade follows its curvature and enters the concave side of the returning blade, eventually slowing down and causing a stagnation pressure, shown in green in the pressure contour plot. This partially counteracts the high pressure at the convex side of the returning blade,

thus mitigating the opposing torque. These effects explain the peak in torque production in the intended direction of rotation when the blade is at this angular position. In summary, the source of torque production at low TSR is 1) partly from the stagnation of a stream of fast flowing air entering the concave side of the advancing blade, and, 2) partly from the lift produced by fast flowing air over the convex side of the advancing blade.

6.1.3 THE SOURCE OF TORQUE PRODUCTION AT HIGH TSR

Once again referring to the c_p vs. TSR plot for turbine position of $x = 1R$, $y = 2.0$ m highlighted in Figure 92, it is noted that there is another local c_p peak at TSR of 1.2. Once again, the turbine torque over one revolution is plotted in Figure 94. In this case, the point of lowest magnitude of torque similarly corresponds to the turbine angular position where the concave side of the advancing blade faces almost perpendicularly to the oncoming wind. The pressure contour plot on the top left of Figure 94 shows a very slight stagnation of the wind in the concave side of the advancing blade, shown in light green. Bear in mind that the TSR is now much higher and the blade is moving at a speed closer to wind speed, hence the stagnation effect is mild. On the convex side of the advancing blade, the low pressure region near the tip (shown in dark blue) is smaller in size compared to the case of TSR 0.8 due to the fact that the blade tip is nearly at the same speed as the wind leading to a smaller a separation bubble, evident in the velocity contour plot at the bottom left of Figure 94. The combination of reduced stagnation on the concave side and reduced low pressure bubble reduces the torque magnitude to zero from 0.1 kNm previously. The opposing torque of the returning blade is similar to the previous case, being caused by severe stagnation on the convex side.

The turbine angular position that gives the greatest torque production is when the concave side of the advancing blade is in a slightly pitched down attitude, i.e., the concave opening is completely shielded from the oncoming wind as shown in the contour plots on the right side of Figure 94. This is surprising given that the Savonius turbine has been labelled as a drag-based device. On the concave side of the advancing blade, being shielded from the oncoming wind, stagnation is generally absent. This is shown by the velocity contour plot on the bottom right of Figure 94. On the convex side,

Figure 94. Torque over one revolution, TSR 1.2.

a large region of low pressure, high speed wind is present, shown by the dark blue colour in pressure plot and red colour in the velocity plot. In the absence of any large pressure differential on both surfaces of the returning blade (shown by approximately the same colour in the pressure plot), it must be concluded that the high torque in the intended direction of rotation is produced purely by the lift above the advancing blade.

Comparing the two torque plots, it is seen that the difference between them is the disappearance of the second torque peak at 2.35 revolutions for the TSR 1.2 case. In summary, the source of torque production at high TSR is purely due to the lift produced by fast flowing air over the convex side of the advancing blade.

6.1.4 DISCUSSION ON OPTIMAL TURBINE AXIS POSITION

From the torque plot of TSR 0.8 in Figure 93, the average torque over one revolution is 0.39 kNm, while that from the torque plot of TSR 1.2 in Figure 94 is 0.30 kNm. However, since the turbine rotating speed is 50% higher in the latter case, it is easily seen that maximum turbine power and hence c_P occurs at the higher TSR. Additionally, it appears that the lift torque remains present at similar magnitude of about 0.6 kNm at both low and high TSRs, but the desirable drag torque disappears at high TSR. Therefore, the optimal turbine axis position would be one that favours torque production by lift, even if it means forgoing desirable drag, since the loss in drag can be more than compensated by lift simply by allowing the turbine to operate at a sufficiently high TSR to produce the required compensation.

The following analysis of the flow field of other turbine positions explains why the line of peak c_P is related to the line of lowest wind speed as shown in Figure 91. Figure 95 to Figure 98 shows the torque plots at peak c_P of all turbine positions. Background observations are:

- For the cases of $x = 1R$ to $3R$, peak c_P occurs at the higher TSR of about 1.2 to 1.4. For the cases of $x = 4R$, peak c_P occurs at the lower TSR of 0.8.
- With the exception of $x = 1R$, $y = 1.5$ m (lowest and front-most position) and $x = 4R$, $y = 3.0$ m, the peak torque is due to lift (corresponding to turbine angular position of about 2.25 revolutions) and its magnitude is quite constant at about 0.6 to 0.7 kNm.
- At $x = 1R$, $y = 1.5$ m, the turbine is largely in the "shadow" of the bluff body step and hence the torque is much lower.
- At $x = 4R$, $y = 3.0$ m, because it is at TSR of 0.8, drag contributes to desirable torque, and causes the maximum torque region to have the characteristic broad shape.

Looking at the flow fields when minimum torque occurs, it is seen that at lower turbine axis vertical positions, there is less opposing pressure caused by stagnation of wind against the convex surface of the returning blade. However, there is also less exposure of the advancing blade to the high wind speed zone which causes a low pressure separation at the tip of the blade and produces desirable torque (Figure 96 left side pressure plots illustrate this observation most obviously). The optimum vertical position is the best compromise between these two effects, resulting in the least severe minimum torque. Near the front of the bluff body edge, for example at $x = 2R$, because the variation of wind speed with vertical direction is drastic (refer to Figure 91), it would be logical to deduce that the optimum position would be where the returning blade is generally near the line of minimum wind speed and where the advancing blade would automatically be immersed in the high wind speed zone. Further away from the bluff body edge, for example at $x = 3R$ and $4R$, the variation of wind speed with vertical direction is more gradual, and if the turbine is located where the advancing blade is in the high speed wind zone, the returning blade would be opposed by a wind speed that is still relatively high. Thus to achieve least severity of minimum torque, it would be logical to deduce that the turbine position has to be nearer the line of minimum wind speed rather than at the midpoint between the lines of maximum and minimum wind speed.

Now looking at the flow fields when maximum torque occurs, two effects are seen. Firstly, as the turbine axis vertical position increases, there is greater diversion of the high speed wind below the returning blade. This causes a mild stagnation pressure ahead of the returning blade thus producing opposing torque (all the right side pressure plots illustrate this effect). Additionally, further from the front of the bluff body edge, for example at $x = 3R$ and $4R$, it interrupts the momentum of the high speed wind above the advancing blade and reduces the size of the blue low pressure region (illustrated by the right side pressure plots of Figure 97 and Figure 98). The second effect is that at higher turbine axis vertical position, the width of the high torque region increases due to the advancing blade spending more time in the high wind speed region. The optimum position is thus the best compromise between the two effects. At positions further from the front edge, due to the additional effect of interruption of wind momentum, avoiding the diversion of high speed wind has greater relative importance than allowing the

Figure 95. Torque plots, pressure & velocity contour plots for turbine position $x = 1R$, y varying from 1.5 m (bottom) to 2.5 m (top).

Figure 96. Torque plots, pressure & velocity contour plots for turbine position $x = 2R$, y varying from 2.0 m (bottom) to 3.0 m (top).

Figure 97. Torque plots, pressure & velocity contour plots for turbine position $x = 3R$, y varying from 2.5 m (bottom) to 3.5 m (top).

Figure 98. Torque plots, pressure & velocity contour plots for turbine position $x = 4R$, y varying from 2.5 m (bottom) to 3.5 m (top).

advancing blade to spend more time in the high wind speed region. Thus to achieve maximum desirable torque, it would be logical to deduce that the turbine position has to be nearer the line of minimum wind speed rather than at the midpoint between the lines of maximum and minimum wind speed.

In summary, the effects determining torque production are:

- Extent and duration of exposure of the advancing blade to the high speed wind zone
- Severity of stagnation of the convex surface of the returning blade
- Influence on the flow field due to the turbine's position

The line of optimal turbine position is a result of the interaction among the above three effects. The empirical observation is that the line has a shape similar to the line of minimum wind speed but shifted by $1.1R$ above it. If a more fundamental explanation based on physical principles for the shape of the line is desired, each of the effects above may be isolated and their relationship with torque studied. However, such a study is beyond the scope of the present thesis.

6.2 NORMALISED EQUATION OF THE LINE OF OPTIMAL INSTALLATION POSITION

The flow field around bluff body forming a forward facing step is well studied and is thus useful as a reference for expressing the line of optimal installation position of a Savonius turbine above the bluff body. The flow field consists of a flow separation bubble after the front edge of the step. With the flow field divided by vertical lines (parallel to the front face of the bluff body), the points of maximum, minimum and free stream wind speed on each vertical line can be joined to form lines of maximum, minimum and free stream wind speed, respectively, as shown in Figure 73 to Figure 75 in section 5.5.1. Using the case with bluff body step height of 5 m, the line of minimum wind speed follows this second-order polynomial:

$$y = -0.11x^2 + 0.38x + 0.074 \quad (6.1)$$

where: y = vertical position above the top surface of the bluff body

x = horizontal position from the front edge of the bluff body

both variables being expressed in terms of H , the bluff body step height.

The applicable Reynolds number using step height as the characteristic length is 3.88×10^6 , while the Reynolds number based on turbine diameter and free stream wind speed is 1.94×10^6 . Since the line of peak c_p lies $1.1R$ above this line of minimum wind speed. Hence:

$$y_{ta} = -0.11x_{ta}^2 + 0.38x_{ta} + 0.074 + 1.1R/H \quad (6.2)$$

where: R = radius of turbine

y_{ta} = vertical position of the turbine axis from the top surface of the bluff body

x_{ta} = horizontal position of the turbine axis from the front edge of the bluff body

The applicable turbine radius is 1.25 m, or $0.25H$. When the turbine is installed in accordance to eq. 6.2, its c_p ranges from 0.5 to 0.6.

6.3 COMPARISON WITH BASELINES

Using settings similar to those used for simulating a turbine above a bluff body, simulation of a turbine above a flat surface was performed. For the horizontal axis cases the vertical position of the turbine axis above the flat surface was 2.5 m, chosen to match the bluff body case that has the best c_p . Two rotating directions were simulated: one similar to the bluff body case where the lower half of the turbine is returning and

Figure 99. Domain for simulation of a turbine above a flat surface, axis horizontal.

Figure 100. For axis horizontal, lower half returning (left) and upper half returning (right).

Figure 101. Domain for simulation of a turbine above a flat surface, axis vertical.

the other where the upper half of the turbine is returning. This is to determine the effect of the flat surface on performance. For the vertical axis case, the turbine is placed asymmetrically near the middle of a sufficiently large domain. The amount of asymmetry follows that used in the validation with Saha's experiment in Section 5.3, i.e., the domain extents on the advancing side and returning side is in the ratio of 5:8. These domains are shown in Figure 99, Figure 100 and Figure 101. Results of the simulation of baseline cases are shown in Figure 102. They agree well with other researchers' results, such as CFD work by Zhou and Rempfer [47] and the list of experimental work in Saha's paper

[23], in particular, the work of Sheldahl *et al* [24]. Peak c_p is approximately 0.2, occurring at a TSR of about 1.0. In contrast, installation above a bluff body boosts the c_p to 0.6 at the optimum installation position, which is a 200% improvement.

Figure 102. Baseline results.

CHAPTER 7. CONCLUSION & FUTURE WORK

7.1 CONCLUSION

The present research begins by verifying that a cross-flow drag-based wind turbine installed above a forward facing step bluff body has a higher power output than one conventionally installed above a flat surface using Computational Fluid Dynamics (CFD) simulation. Subsequent to that, the research endeavours to find the line of optimal installation position of a two-bladed Savonius turbine above a bluff body to maximise the coefficient of power (c_p). It was discovered that this line bears the relatively simple relationship of an offset of $1.1R$ (R : turbine radius) above the line of minimum wind speed in the flow field of a forward facing step bluff body. The discovery is based on results obtained from CFD simulation of a range of turbine positions validated by driving test experiments of one installation position at free stream wind speeds from 8 to 12 m/s. When the turbine is installed with its axis on this line, the c_p ranges from 0.53 to 0.60. These numbers compare very favourably with a conventionally installed turbine which has c_p of 0.23. The methodology used can also be applied to further work involving other combinations of parameters.

CFD simulation was performed using the commercial software ANSYS CFX in the Workbench environment which is designed for performing parametric studies conveniently. The simulation domain consists of an infinite-width bluff body with a forward facing step of height five metres, above which is installed a two-bladed Savonius turbine with its axis horizontal and rotating with lower half returning. The turbine axis position is variable horizontally and vertically. The turbine radius is 1.25 metres and its half-width (since it is installed horizontally) is twice its radius. The domain is subdivided into a rotating region with the turbine form represented as voids, a wake region of fine mesh enclosing the rotating region and spanning six turbine diameters downstream and a region representing the surrounding air. The overall domain height is $20H$ (H : step height) while domain length before and after the step is $15H$ and $20H$ respectively. Meshing was performed automatically by ANSYS Meshing with user-control of element edge length to about 4% of turbine diameter at the turbine surfaces, 6% in the wake

region and growing at 1.1 beyond. Typically, the number of elements in the domain is about 5 million.

Simulations were performed for a free stream wind speed of 12 m/s over a range of turbine axis horizontal positions of $0.25H$ to $1.0H$ from the front edge of the step and vertical position of 1.5 to 3.0 metres above the step. For each position, a range of TSR with lower limit of 0.6 to 0.8 and upper limit of 1.3 to 1.6 were simulated. Each simulation begins with a frozen rotor run to quickly obtain an approximate flow field which is used as the initial condition of a following transient run of 2.5 to 3.5 revolutions. The Shear Stress Transport turbulence model was used. The average torque over the final revolution is extracted for analysis and calculation of c_p .

Prior to these simulations, validation of the simulation settings was made using driving test results for the simulation of the turbine and using prior literature for the simulation of flow over the forward facing step. Driving tests were conducted on a turbine of radius 0.55 m and width 1 m installed above a finite-width bluff body with a frontal rectangular face of 2.4 m wide and 1.3 m high. The top surface of the bluff body is approximately 2.5 m above the road surface. The entire test rig is mounted on a lorry which is partially concealed within the bluff body. Wind speed was measured with an acoustic anemometer placed alongside the turbine 2 m to the starboard of the turbine mid-plane and 0.8 m above its axis. Tests were conducted at driving speeds of 8, 10 and 12 m/s at TSRs ranging from 0.5 to 1.4 (using vehicle speed as reference). It was found that experimental runs where the prevailing cross wind was low agree well with simulations.

In preparation for driving tests, both wind tunnel tests and CFD simulation were used to determine the installation position of the wind turbine on the test rig to obtain the best power output. For the wind tunnel experiments, tests on a full width forward facing step of height 0.2 m installed in a test section of width 0.78 m and height 0.72 m to obtain a flow field similar to that expected for a step in an unconstrained environment. A much reduced-scale turbine with diameter 0.1 m was installed at a fixed vertical position of $1.2R$ above the top of the step at various horizontal positions and loaded with constant friction. It was found that forward most position gave the highest turbine speed. However, for practical reasons, the turbine on the test rig cannot protrude ahead of the

front face of the bluff body. CFD simulations using default settings to find the most powerful installation position, on the other hand, showed that the best horizontal position is where the turbine axis is $1R$ behind the front face of the bluff body, i.e., the circumference of the turbine is tangent to the front face. Considering both outcomes, the turbine on the test rig was installed at a vertical position of $1.5R$ above the top of the bluff body and $1R$ behind the front face for driving tests.

From a broader perspective, the methodology used in the present research can be used in any new situation involving different combinations of parameters such as building dimensions, turbine size, etc., to similarly determine the optimum installation position of a wind turbine on top of a building. The improvement in performance by employing this concept can mitigate the poor performance of urban wind turbines compared to their large scale counterparts installed in rural or offshore wind farms.

7.2 FUTURE WORK

7.2.1 CFD SIMULATION WORK

The simple empirical formula describing the line of optimal installation position is obtained based on CFD simulation results of a turbine of diameter 2.5 metres, exposed to one wind speed of 12 m/s and installed above an infinitely wide bluff body of height five metres. Therefore, future CFD simulation work should fill knowledge gaps in the understanding of turbine behaviours over a range of: i) turbine sizes, ii) wind speeds, and iii) bluff body step heights.

It is the author's opinion that the aspect of turbine sizes is of highest importance. The analysis of optimal turbine position in section 6.1.4 is based on the optimal balance of the opposing effects of the advancing and returning blade in different wind speed zones. With a change in turbine diameter, there is a possibility that the point of optimal balance may be shifted, perhaps also with some correlation to turbine diameter. Furthermore, a range of turbine sizes will be definitely encountered in practical applications arising from energy needs and budget considerations pertinent to each case. Hence a more generalised knowledge that covers a range of turbine sizes is most urgently needed to supplement the current knowledge.

Second in priority would be the aspect of wind speeds. At any potential wind turbine installation site, the wind condition is certainly not constant but varies with time. Thus more comprehensive knowledge covering a range of wind speeds is also needed for the prediction of energy production when the distribution of wind speeds at a proposed installation site is known.

Thirdly, it is reasonable to expect that the bluff body step height would vary from location to location, be it above a man-made building or above a natural topographical feature such as a cliff. The present research includes CFD simulation of flow over bluff bodies of various step heights from five to fifteen metres. It was found that when normalised using the step height, the line of minimum wind speed for the three cases are nearly identical. If future simulation work on a range of turbine sizes finds that it does not affect the line of optimal installation position, then it is likely that the result for different bluff body step heights would also be unchanged, since both approaches have same effect of varying the size of the turbine relative to the features of the flow field.

Finally, the flow field may be different in the case of a finite-width bluff body, such as a building, thus influencing turbine behaviour. Future simulation should also look into this effect.

7.2.2 EXPERIMENTAL WORK

CFD simulation results were validated with results from driving test experiments on only a single turbine installation position. Due to limited experimental resources, further driving tests were not performed covering other installation positions to re-confirm the validity of the empirical formula for optimal installation position. It is thus proposed that future experimental work attempt to fill this shortcoming, by performing driving tests with the turbine at varying positions.

Another area that deserves future experimentation is the confirmation of the principle of torque production that was discussed in Sections 0 and 6.1.3. To do that, instantaneous torque measurement capability together linked with turbine angular position measurement has to be provided in the apparatus. A further enhancement will

be the capability to measure pressure on the surface of the turbine blade using suitable pressure tapings.

In both the above suggested experimental work, the researcher is advised to follow the procedural guidelines in Appendix B for safety and work effectiveness.

REFERENCES

- [1] Kema, Inc., "Renewable Energy Cost of Generation Update," California Energy Commission, California, 2009.
- [2] J. E. Bos, "Feasibility Study of the WindNok," TU Delft, Delft, 2010.
- [3] Observ'ER, "Worldwide Electricity Production from Renewable Sources," L'Observatoire des énergies renouvelables, Paris, 2012.
- [4] Bloomberg New Energy Finance, "Onshore Wind to Reach Parity with Fossil-Fuel Electricity by 2016," Bloomberg New Energy Finance, New York, 2011.
- [5] Wineur, "Guidelines for Small Wind Turbines in the Built Environment," European Cities Urban Wind Network, Lyon, 2007.
- [6] S. C. van Beveren, "Design of an Urban Wind Turbine with Diffuser," TU Delft, Delft, 2008.
- [7] B. M. van Wijk, "Predicting the Rooftop Wind Climate for Urban Wind Energy in the Rotterdam - Delft - Zoetermeer Region," TU Delft, Delft, 2011.
- [8] Joe, "Here goes (please comment)," 11 April 2007. [Online]. Available: <http://archinect.com/blog/article/21451130/here-goes-please-comment>. [Accessed 19 June 2012].
- [9] "Falcon Vertical Axis Wind Turbine," 2012. [Online]. Available: <http://www.wepower.us/products/falcon/index.html>. [Accessed 19 June 2012].

- [10] Schlaich, Bergermann und Partner, "Solar Updraft Tower," 2012. [Online]. Available: http://www.sbp.de/en/sun/sheet/82-Solar_Chimney_Pilot_Plant_Manzanares.pdf. [Accessed 20 June 2012].
- [11] "Darrieus Wind Turbine," 12 July 2008. [Online]. Available: http://upload.wikimedia.org/wikipedia/commons/d/d7/Eoliennes_Gaspesie.jpg. [Accessed 20 June 2012].
- [12] "Photos of Vertical Axis Wind Turbines by Paul Gipe," 2012. [Online]. Available: <http://www.wind-works.org/photos/PhotosVAWTs.html>. [Accessed 20 June 2012].
- [13] "Quiet Revolution qr5 Wind Turbine," 2012. [Online]. Available: <http://www.quietrevolution.com/qr5-turbine.htm>. [Accessed 20 June 2012].
- [14] "Windside Wind Turbine," 2012. [Online]. Available: <http://www.windside.com/products>. [Accessed 20 June 2012].
- [15] "Home Page of High Tech GenSets," 2012. [Online]. Available: <http://hightechgensets.com/>. [Accessed 20 June 2012].
- [16] "Ecowing Wind Turbine," 2012. [Online]. Available: <http://www.chiba-machinery.co.jp/e/ew/index.html>. [Accessed 20 June 2012].
- [17] "Hi-VAWT DS-3000W Wind Turbine," 2012. [Online]. Available: <http://www.hi-vawt.com.tw/en/ds3000w.html>. [Accessed 20 June 2012].
- [18] "VAWT Page," 28 August 2008. [Online]. Available: <http://www.windstuffnow.com/main/vawt.htm>. [Accessed 20 June 2012].
- [19] "Cygnus Power Wind Turbine," 2012. [Online]. Available: <http://www.cygnus->

power.com/products. [Accessed 20 June 2012].

- [20] A. Betz, "Das maximum der theoretisch möglichen ausnützung des windes durch windmotoren," *Zeitschrift f. d. gesamte Turbinenwesen*, vol. 26, 1920.
- [21] F. M. White, *Fluid Mechanics*, New York: McGraw-Hill, 2011.
- [22] R. Noll, N. D. Ham, H. M. Drees and N. L. B., "1 kW High Reliability Wind System Development," United States Department of Energy, Rocky Flats, 1982.
- [23] U. K. Saha and M. J. Rajkumar, "On the performance Analysis of Savonius Rotor with Twisted Blades," *Renewable Energy*, vol. 31, pp. 1776-1788, 2006.
- [24] B. F. Blackwell, R. E. Sheldahl and L. V. Feltz, "Wind Tunnel Performance Data for Two- and Three-Bucket Savonius Rotors," Sandia Laboratories, Albuquerque, 1977.
- [25] J. Y. Park, S. Lee, T. Sabourin and K. Park, "OSEA Archives," 2012. [Online]. Available: http://ontario-sea.org/Storage/27/1843_A_Novel_Vertical-Axis_Wind_Turbine_for_Distributed_%26_Utility_Deployment.pdf. [Accessed 22 June 2012].
- [26] M. Takao, K. Hideki, T. Maeda, Y. Kamada, M. Oki and A. Minoda, "A Straight-bladed Vertical Axis Wind Turbine with a Directed Guide Vane Row," vol. 18, no. 1, 2009.
- [27] M. Mohamed, G. Janiga, E. Pap and D. Thévenin, "Optimization of Savonius turbines using an obstacle shielding the returning blade," *Renewable Energy*, vol. 35, pp. 2618-2626, 2010.
- [28] "Honeywell WT6500 Wind Turbine," 2012. [Online]. Available: <http://www.windtronics.com>. [Accessed 22 June 2012].

- [29] "Wind Turbine Big & Small at Windpower 2008," 3 June 2008. [Online]. Available: <http://www.jetsongreen.com/2008/06/wind-turbines-b.html>. [Accessed 22 June 2012].
- [30] "VQ Windjets Product Page," 2011. [Online]. Available: <http://www.aeswind.com/>. [Accessed 22 June 2012].
- [31] B. D. Altan and M. Atilgan, "The use of a curtain design to increase the performance level of a Savonius wind rotor," *Renewable Energy*, vol. 35, pp. 821-829, 2010.
- [32] H. K. Versteeg and W. Malalasekera, *An Introduction to Computational Fluid Dynamics: The Finite Volume Method* 2nd ed., Prentice Hall, 2007.
- [33] S. Murakami, A. Mochida and Y. Hayashi, "Examining the k- ϵ model by means of a wind tunnel test and large eddy simulation around a cube.," *Journal of Wind Engineering and Industrial Aerodynamics*, vol. 35, pp. 87-100, 1990.
- [34] P. Durbin and B. Pettersson Reif, *Statistical Theory and Modeling for Turbulent Flows*, 2nd ed., Chichester: Wiley, 2010.
- [35] F. R. Menter, "Two-equation eddy viscosity turbulence models for engineering applications," *AIAA Journal*, vol. 32, no. 8, pp. 1598-1605, 1994.
- [36] D. C. Wilcox, "Reassessment of the scale-determining equation for advanced turbulence models," *AIAA*, vol. 26, no. 11, pp. 1299-1310, 1988.
- [37] P. E. Smirnov and F. R. Menter, "Sensitization of the SST turbulence model to rotation and curvature by applying the Spalart-Shur correction term," *Journal of Turbomachinery*, vol. 131, no. 4, 2009.

- [38] National Environment Agency, "WeatherWise Singapore," 2009. [Online]. Available: <http://app2.nea.gov.sg/data/cmsresource/20090721544571208250.pdf>. [Accessed 28 May 2012].
- [39] C. L. Murray and P. S. Negi, *Wind Resource Data from Various Locations in Singapore*, Singapore: Unpublished, 2012.
- [40] K. McLaren, S. Tullis and S. Zaida, "Computational fluid dynamics simulation of the aerodynamics of a high solidity, small-scale vertical axis wind turbine," *Wind Engineering*, vol. 15, pp. 349-361, 2012.
- [41] R. Nobile, M. Vahdati, J. Barlow and A. Mewburn-Crook, "Dynamic Stall For A Vertical Axis Wind Turbine In A Two-Dimensional Study," in *Technologies for Sustainable Built Environments Engineering Doctorate Conference*, Reading, 2011.
- [42] S. Baker, "Regions of Recirculating Flow Associated with Two-dimensional Steps," University of Surrey, Guildford, 1977.
- [43] S. Larwood, J. Sencenbaugh and B. Acker, "Controlled Velocity Testing of an 8 kW Wind Turbine," in *WindPower 2001 Conference*, Washington DC, 2001.
- [44] B. Feeny and D. Liu, "Wind Turbine Power Measurement Test Rig at MSU," 2010. [Online]. Available: <http://researchgroups.msu.edu/vawt/projects/wind-turbine-power-measurement-test-rig-msu>. [Accessed 22 June 2012].
- [45] J. L. Menet, "A Double-step Savonius rotor for local production of electricity: a design study," *Renewable Energy*, vol. 29, pp. 1843-1862, 2004.
- [46] M. Persson, "GLONASS support in our latest Xperia phones," 19 January 2012. [Online]. Available: <http://developer.sonymobile.com/2012/01/19/glonass-support->

in-our-latest-xperia-phones/. [Accessed 30 Jun 2014].

- [47] T. Zhou and D. Rempfer, "Numerical study of detailed flow field and performance of Savonius wind turbines," *Renewable Energy*, vol. 51, pp. 373-381, 2013.

APPENDIX A

CAD files of test apparatus, simulation result files, and spreadsheets of processed results are available upon request. Contact the author at gohsc_ernest@yahoo.com.sg

APPENDIX B

The month-long driving test season allowed the experimental team members including the author to gain experience with practical considerations of such work. They are:

- Test rig and turbine should be secured by bolts to the chassis of the vehicle. Initially, the test rig was merely tied down to the roof and load bed of the lorry. After just two hours, the installation had come loose. The driving test had to be suspended for two days in order to make modifications to the rig, the most important addition of which was an added strut bolted to the front of the lorry's ladder frame (above the bumper) supporting the front of the test rig. Other additions include wire ropes secured with eyebolts to the load bed of the lorry for positively tying the rig down. In any future driving test, it is recommended that the vehicle be selected early, the available mounting holes on the chassis be mapped out and the test rig be designed to utilise the holes.
- Establish a safety inspection procedure before setting off for each test that is to be strictly followed. On the very first day of driving test, one side of the turbine bearing block came loose. It was later discovered that the T-nuts securing it to the extruded aluminium profile were not properly crossed in their slots. In any future tests, a critical evaluation of the risks pertaining to components dislodging or loosening must be carried out and the inspection procedure to arrest such risks must be drawn up and entered into the University's risk assessment form. Since such tests will likely to be conducted in the early hours of the morning in the dark, adequate lighting and visual access for the inspector must be provided. The leader of the team must instil a genuine safety mind-set in his members and dispel any thinking that the University's risk management policies are mere procedures to which only lip service need to be paid.
- Cater for field adjustments of the rig to compensate for pitch and roll when at the actual test site. The lorry's load bed is not perfectly level in pitch except when loaded with a particular weight. In future driving tests whose purpose is to validate selected cases of simulations involving a perfectly level bluff body instead of merely validating simulation settings, the ability to exactly replicate what was simulated is important.

Hence the need for field adjustable pitch levelling. The road surface is cambered (middle of the road higher than the side) and to make the effects of cross wind simpler to measure, roll adjustment may also be provided.

- Provide means of directly measuring the free stream wind speed. As shown in Section 4.5.9 Driving Test Results and Discussion, the vehicle speed is not an accurate indicator of free stream wind speed due to prevailing wind and disturbance from neighbouring vehicles. Therefore a means of measuring the free stream wind speed seen by the turbine should be provided. A possible means is a multi-hole pitot tube supported by a sufficiently long spar ahead of the vehicle.
- Use electrical instead of mechanical braking. The mechanical brake used in the present research, in spite of being actuated by a motor with digital pulse width modulation, was a major problem that took a lot of time to resolve. Repeatability and fine variation of braking torque was difficult to achieve. Although marginally satisfactory performance was eventually achieved by applying gear oil to the brake disc, any future driving test must adopt an electrical braking method such as a generator with an accurately controllable load or a suitably sized magnetic particle brake.
- Provide an independent electrical supply for instrumentation. Initially, a lot of problems with noisy results captured on the PC-based oscilloscopes were encountered. This was due to the highly noisy electrical system of the vehicle. After trying various means of noise isolation, eventually a separate lead-acid accumulator was used to provide electrical power to all instruments. In future experiments, instruments should preferably run on convenient low voltage DC, e.g., 12 volts so that they can be powered from an accumulator. Proper noise shielding of signal-carrying wires should also be provided.
- Schedule intermissions in testing. As previously mentioned, testing was suspended after the first test to rectify problems. In future work, a recommended schedule would be for the initial test to be scheduled on Tuesday morning (i.e., after midnight on Monday) so that the working day before can be used for various preparatory activities. The following day (Wednesday) should not be scheduled as a test day so that review of intermediate results, adjustments to apparatus, etc., can be made. The

test schedule should only be made consecutive when apparatus and procedures have stabilised. Even then, consecutive test days should be confined to Tuesday to Thursday, leaving Friday as a working day for the review of results and adjustment of the following week's test plan, if required.

- Re-commission the gantry crane in the mechanical workshop for handling the turbine and rig. In the present research, a lorry crane was hired on two occasions, once to hoist the test rig on the lorry and a second time, to hoist it off. In between, whenever there was a need to make adjustments or improvement to the securing of the rig, manual forklifts and car jacks propped on ladders were used. This was cumbersome, unsafe and time consuming. It is thus recommended that in future, the existing gantry crane in the mechanical workshop be re-commissioned and roped in to cater to all hoisting needs.

APPENDIX C

Simulation settings for determining step height in wind tunnel. (Section 5.1)

Setting up CFX Solver run ...

```
+-----+
|                                     |
|                               CFX Command Language for Run             |
|                                     |
+-----+
```

LIBRARY:

```
MATERIAL: Air at 25 C
  Material Description = Air at 25 C and 1 atm (dry)
  Material Group = Air Data, Constant Property Gases
  Option = Pure Substance
  Thermodynamic State = Gas
PROPERTIES:
  Option = General Material
EQUATION OF STATE:
  Density = 1.185 [kg m^-3]
  Molar Mass = 28.96 [kg kmol^-1]
  Option = Value
END
SPECIFIC HEAT CAPACITY:
  Option = Value
  Specific Heat Capacity = 1.0044E+03 [J kg^-1 K^-1]
  Specific Heat Type = Constant Pressure
END
REFERENCE STATE:
  Option = Specified Point
  Reference Pressure = 1 [atm]
  Reference Specific Enthalpy = 0. [J/kg]
  Reference Specific Entropy = 0. [J/kg/K]
  Reference Temperature = 25 [C]
END
DYNAMIC VISCOSITY:
  Dynamic Viscosity = 1.831E-05 [kg m^-1 s^-1]
  Option = Value
END
THERMAL CONDUCTIVITY:
  Option = Value
  Thermal Conductivity = 2.61E-02 [W m^-1 K^-1]
END
ABSORPTION COEFFICIENT:
  Absorption Coefficient = 0.01 [m^-1]
  Option = Value
END
SCATTERING COEFFICIENT:
  Option = Value
  Scattering Coefficient = 0.0 [m^-1]
END
REFRACTIVE INDEX:
  Option = Value
  Refractive Index = 1.0 [m m^-1]
END
THERMAL EXPANSIVITY:
  Option = Value
  Thermal Expansivity = 0.003356 [K^-1]
END
END
END
FLOW: Flow Analysis 1
SOLUTION UNITS:
  Angle Units = [rad]
  Length Units = [m]
  Mass Units = [kg]
  Solid Angle Units = [sr]
  Temperature Units = [K]
  Time Units = [s]
END
```

```

ANALYSIS TYPE:
  Option = Steady State
EXTERNAL SOLVER COUPLING:
  Option = None
END
END
DOMAIN: Default Domain
  Coord Frame = Coord 0
  Domain Type = Fluid
  Location = B72
BOUNDARY: Default Domain Default
  Boundary Type = WALL
  Location = F64.72,F66.72,F67.72,F68.72,F70.72,F71.72
BOUNDARY CONDITIONS:
  MASS AND MOMENTUM:
    Option = No Slip Wall
  END
  WALL ROUGHNESS:
    Option = Smooth Wall
  END
END
END
BOUNDARY: Inlet
  Boundary Type = INLET
  Location = F69.72
BOUNDARY CONDITIONS:
  FLOW REGIME:
    Option = Subsonic
  END
  MASS AND MOMENTUM:
    Normal Speed = 8 [m s-1]
    Option = Normal Speed
  END
  TURBULENCE:
    Option = Low Intensity and Eddy Viscosity Ratio
  END
END
END
BOUNDARY: Outlet
  Boundary Type = OPENING
  Location = F65.72
BOUNDARY CONDITIONS:
  FLOW DIRECTION:
    Option = Normal to Boundary Condition
  END
  FLOW REGIME:
    Option = Subsonic
  END
  MASS AND MOMENTUM:
    Option = Opening Pressure and Direction
    Relative Pressure = 0 [Pa]
  END
  TURBULENCE:
    Option = Zero Gradient
  END
END
END
DOMAIN MODELS:
  BUOYANCY MODEL:
    Option = Non Buoyant
  END
  DOMAIN MOTION:
    Option = Stationary
  END
  MESH DEFORMATION:
    Option = None
  END
  REFERENCE PRESSURE:
    Reference Pressure = 1 [atm]
  END
END
FLUID DEFINITION: Fluid 1
  Material = Air at 25 C
  Option = Material Library
  MORPHOLOGY:
    Option = Continuous Fluid

```



```

END
END
FLUID MODELS:
  COMBUSTION MODEL:
    Option = None
  END
  HEAT TRANSFER MODEL:
    Option = None
  END
  THERMAL RADIATION MODEL:
    Option = None
  END
  TURBULENCE MODEL:
    Option = k epsilon
  END
  TURBULENT WALL FUNCTIONS:
    Option = Scalable
  END
END
INITIALISATION:
  Option = Automatic
  INITIAL CONDITIONS:
    Velocity Type = Cartesian
    CARTESIAN VELOCITY COMPONENTS:
      Option = Automatic with Value
      U = 6 [m s-1]
      V = 0 [m s-1]
      W = 0 [m s-1]
    END
    STATIC PRESSURE:
      Option = Automatic with Value
      Relative Pressure = 0 [Pa]
    END
    TURBULENCE INITIAL CONDITIONS:
      Option = Low Intensity and Eddy Viscosity Ratio
    END
  END
END
END
OUTPUT CONTROL:
  RESULTS:
    File Compression Level = Default
    Option = Standard
  END
END
SOLVER CONTROL:
  Turbulence Numerics = First Order
  ADVECTION SCHEME:
    Option = High Resolution
  END
  CONVERGENCE CONTROL:
    Length Scale Option = Conservative
    Maximum Number of Iterations = 100
    Minimum Number of Iterations = 1
    Timescale Control = Auto Timescale
    Timescale Factor = 1.0
  END
  CONVERGENCE CRITERIA:
    Residual Target = 0.001
    Residual Type = RMS
  END
  DYNAMIC MODEL CONTROL:
    Global Dynamic Model Control = On
  END
END
END

```

Simulation settings for determining optimum turbine position of driving test rig. (Section 5.2)

Setting up CFX Solver run ...

```
+-----+
|                                             |
|               CFX Command Language for Run   |
|                                             |
+-----+
```

LIBRARY:

CEL:

EXPRESSIONS:

```
Free Stream Wind Speed = 12[m s^-1]
Rotating Speed = TSR[rad]*Free Stream Wind Speed /Turbine Radius
TSR = 1.1
Time Step = Time for 1 Rev /120
Time for 1 Rev = 2*pi[rad]/Rotating Speed
Time for 3 Revs = 3*Time for 1 Rev
Time for 6 Revs = 6*Time for 1 Rev
Time for 9 Revs = 9*Time for 1 Rev
Turbine Radius = 1.1 [m]
```

END

END

MATERIAL: Air at 25 C

```
Material Description = Air at 25 C and 1 atm (dry)
Material Group = Air Data, Constant Property Gases
Option = Pure Substance
Thermodynamic State = Gas
```

PROPERTIES:

Option = General Material

EQUATION OF STATE:

```
Density = 1.185 [kg m^-3]
Molar Mass = 28.96 [kg kmol^-1]
Option = Value
```

END

SPECIFIC HEAT CAPACITY:

```
Option = Value
Specific Heat Capacity = 1.0044E+03 [J kg^-1 K^-1]
Specific Heat Type = Constant Pressure
```

END

REFERENCE STATE:

```
Option = Specified Point
Reference Pressure = 1 [atm]
Reference Specific Enthalpy = 0. [J/kg]
Reference Specific Entropy = 0. [J/kg/K]
Reference Temperature = 25 [C]
```

END

DYNAMIC VISCOSITY:

```
Dynamic Viscosity = 1.831E-05 [kg m^-1 s^-1]
Option = Value
```

END

THERMAL CONDUCTIVITY:

```
Option = Value
Thermal Conductivity = 2.61E-02 [W m^-1 K^-1]
```

END

ABSORPTION COEFFICIENT:

```
Absorption Coefficient = 0.01 [m^-1]
Option = Value
```

END

SCATTERING COEFFICIENT:

```
Option = Value
Scattering Coefficient = 0.0 [m^-1]
```

END

REFRACTIVE INDEX:

```
Option = Value
Refractive Index = 1.0 [m m^-1]
```

END

THERMAL EXPANSIVITY:

```
Option = Value
Thermal Expansivity = 0.003356 [K^-1]
```

END

```

END
END
END
FLOW: Flow Analysis 1
  SOLUTION UNITS:
    Angle Units = [rad]
    Length Units = [m]
    Mass Units = [kg]
    Solid Angle Units = [sr]
    Temperature Units = [K]
    Time Units = [s]
  END
  ANALYSIS TYPE:
    Option = Transient
  EXTERNAL SOLVER COUPLING:
    Option = None
  END
  INITIAL TIME:
    Option = Automatic with Value
    Time = 0 [s]
  END
  TIME DURATION:
    Option = Total Time
    Total Time = Time for 9 Revs
  END
  TIME STEPS:
    Option = Timesteps
    Timesteps = Time Step
  END
END
DOMAIN: Rotating Region
  Coord Frame = Coord 0
  Domain Type = Fluid
  Location = Assembly
  BOUNDARY: Domain Interface 1 Side 2
    Boundary Type = INTERFACE
    Location = Interface RR
  BOUNDARY CONDITIONS:
    MASS AND MOMENTUM:
      Option = Conservative Interface Flux
    END
    TURBULENCE:
      Option = Conservative Interface Flux
    END
  END
END
BOUNDARY: Symmetry RR
  Boundary Type = SYMMETRY
  Location = Symmetry RR
END
BOUNDARY: Turbine Surfaces
  Boundary Type = WALL
  Frame Type = Rotating
  Location = Wall Turbine Surfaces
  BOUNDARY CONDITIONS:
    MASS AND MOMENTUM:
      Option = No Slip Wall
    END
    WALL ROUGHNESS:
      Option = Smooth Wall
    END
  END
END
DOMAIN MODELS:
  BUOYANCY MODEL:
    Option = Non Buoyant
  END
  DOMAIN MOTION:
    Angular Velocity = -Rotating Speed
    Option = Rotating
  AXIS DEFINITION:
    Option = Coordinate Axis
    Rotation Axis = Coord 0.3
  END
END
MESH DEFORMATION:

```

```

    Option = None
END
REFERENCE PRESSURE:
    Reference Pressure = 1 [atm]
END
END
FLUID DEFINITION: Fluid 1
    Material = Air at 25 C
    Option = Material Library
MORPHOLOGY:
    Option = Continuous Fluid
END
END
FLUID MODELS:
    COMBUSTION MODEL:
        Option = None
    END
    HEAT TRANSFER MODEL:
        Option = None
    END
    THERMAL RADIATION MODEL:
        Option = None
    END
    TURBULENCE MODEL:
        Option = k omega
    END
    TURBULENT WALL FUNCTIONS:
        Option = Automatic
    END
END
INITIALISATION:
    Frame Type = Stationary
    Option = Automatic
    INITIAL CONDITIONS:
        Velocity Type = Cartesian
        CARTESIAN VELOCITY COMPONENTS:
            Option = Automatic with Value
            U = Free Stream Wind Speed
            V = 0 [m s-1]
            W = 0 [m s-1]
        END
        STATIC PRESSURE:
            Option = Automatic with Value
            Relative Pressure = 0 [Pa]
        END
        TURBULENCE INITIAL CONDITIONS:
            Option = Medium Intensity and Eddy Viscosity Ratio
        END
    END
END
END
DOMAIN: Surrounding Air
    Coord Frame = Coord 0
    Domain Type = Fluid
    Location = Assembly 2
BOUNDARY: Domain Interface 1 Side 1
    Boundary Type = INTERFACE
    Location = Interface SA
    BOUNDARY CONDITIONS:
        MASS AND MOMENTUM:
            Option = Conservative Interface Flux
        END
        TURBULENCE:
            Option = Conservative Interface Flux
        END
    END
END
END
BOUNDARY: Inlet
    Boundary Type = INLET
    Location = Inlet
    BOUNDARY CONDITIONS:
        FLOW REGIME:
            Option = Subsonic
        END
        MASS AND MOMENTUM:
            Normal Speed = Free Stream Wind Speed

```

```

        Option = Normal Speed
    END
    TURBULENCE:
        Option = Medium Intensity and Eddy Viscosity Ratio
    END
END
BOUNDARY: Opening
    Boundary Type = OPENING
    Location = Opening Top
    BOUNDARY CONDITIONS:
        FLOW DIRECTION:
            Option = Normal to Boundary Condition
        END
        FLOW REGIME:
            Option = Subsonic
        END
        MASS AND MOMENTUM:
            Option = Opening Pressure and Direction
            Relative Pressure = 0 [Pa]
        END
        TURBULENCE:
            Option = Medium Intensity and Eddy Viscosity Ratio
        END
    END
END
BOUNDARY: Outlet
    Boundary Type = OUTLET
    Location = Outlet
    BOUNDARY CONDITIONS:
        FLOW REGIME:
            Option = Subsonic
        END
        MASS AND MOMENTUM:
            Option = Average Static Pressure
            Pressure Profile Blend = 0.05
            Relative Pressure = 0 [Pa]
        END
        PRESSURE AVERAGING:
            Option = Average Over Whole Outlet
        END
    END
END
BOUNDARY: Symmetry Far Field
    Boundary Type = SYMMETRY
    Location = Symmetry Far Field
END
BOUNDARY: Symmetry SA
    Boundary Type = SYMMETRY
    Location = Symmetry SA
END
BOUNDARY: Wall Floor
    Boundary Type = WALL
    Location = Wall Floor
    BOUNDARY CONDITIONS:
        MASS AND MOMENTUM:
            Option = No Slip Wall
        END
        WALL ROUGHNESS:
            Option = Smooth Wall
        END
    END
END
DOMAIN MODELS:
    BUOYANCY MODEL:
        Option = Non Buoyant
    END
    DOMAIN MOTION:
        Option = Stationary
    END
    MESH DEFORMATION:
        Option = None
    END
    REFERENCE PRESSURE:
        Reference Pressure = 1 [atm]
    END
END

```

```

END
FLUID DEFINITION: Fluid 1
  Material = Air at 25 C
  Option = Material Library
MORPHOLOGY:
  Option = Continuous Fluid
END
END
FLUID MODELS:
  COMBUSTION MODEL:
    Option = None
  END
  HEAT TRANSFER MODEL:
    Option = None
  END
  THERMAL RADIATION MODEL:
    Option = None
  END
  TURBULENCE MODEL:
    Option = k omega
  END
  TURBULENT WALL FUNCTIONS:
    Option = Automatic
  END
END
INITIALISATION:
  Option = Automatic
  INITIAL CONDITIONS:
    Velocity Type = Cartesian
    CARTESIAN VELOCITY COMPONENTS:
      Option = Automatic with Value
      U = Free Stream Wind Speed
      V = 0 [m s-1]
      W = 0 [m s-1]
    END
    STATIC PRESSURE:
      Option = Automatic with Value
      Relative Pressure = 0 [Pa]
    END
    TURBULENCE INITIAL CONDITIONS:
      Option = Medium Intensity and Eddy Viscosity Ratio
    END
  END
END
DOMAIN INTERFACE: Domain Interface 1
  Boundary List1 = Domain Interface 1 Side 1
  Boundary List2 = Domain Interface 1 Side 2
  Interface Type = Fluid Fluid
INTERFACE MODELS:
  Option = General Connection
FRAME CHANGE:
  Option = Transient Rotor Stator
END
MASS AND MOMENTUM:
  Option = Conservative Interface Flux
MOMENTUM INTERFACE MODEL:
  Option = None
END
END
PITCH CHANGE:
  Option = None
END
END
MESH CONNECTION:
  Option = GGI
END
END
OUTPUT CONTROL:
  RESULTS:
    File Compression Level = Default
    Option = Standard
  END
TRANSIENT RESULTS: Transient Results 1
  Extra Output Variables List = Velocity in Stn Frame, Velocity in Stn \
    Frame u, Velocity in Stn Frame v, Velocity in Stn Frame w

```

```

File Compression Level = Default
Option = Smallest
OUTPUT FREQUENCY:
    Option = Timestep Interval
    Timestep Interval = 10
END
END
TRANSIENT RESULTS: Transient Results 2
    Extra Output Variables List = Velocity in Stn Frame,Velocity in Stn \
        Frame u,Velocity in Stn Frame v,Velocity in Stn Frame w
    File Compression Level = Default
    Option = Smallest
    OUTPUT FREQUENCY:
        Option = Timestep List
        Timestep List = 962, 964, 966, 968, 970, 972, 974, 976, 978, 980, \
            982, 984, 986, 988, 990, 992, 994, 996, 998, 1000, 1002, 1004, \
            1006, 1008, 1010, 1012, 1014, 1016, 1018, 1020, 1022, 1024, 1026, \
            1028, 1030, 1032, 1034, 1036, 1038, 1040, 1042, 1044, 1046, 1048, \
            1050, 1052, 1054, 1056, 1058, 1060, 1062, 1064, 1066, 1068, 1070, \
            1072, 1074, 1076, 1078, 1080
    END
END
END
SOLVER CONTROL:
    Turbulence Numerics = First Order
    ADVECTION SCHEME:
        Option = Upwind
    END
    CONVERGENCE CONTROL:
        Maximum Number of Coefficient Loops = 10
        Minimum Number of Coefficient Loops = 1
        Timescale Control = Coefficient Loops
    END
    CONVERGENCE CRITERIA:
        Residual Target = 0.001
        Residual Type = RMS
    END
    TRANSIENT SCHEME:
        Option = First Order Backward Euler
    END
END
END
COMMAND FILE:
    Version = 13.0
    Results Version = 13.0
END
PARAMETERIZATION:
    INPUT FIELD: TSR
        Expression Name = TSR
        Method = Expression
    END
END
SIMULATION CONTROL:
    EXECUTION CONTROL:
        EXECUTABLE SELECTION:
            Double Precision = Off
        END
        INTERPOLATOR STEP CONTROL:
            Runtime Priority = Standard
        MEMORY CONTROL:
            Memory Allocation Factor = 1.0
        END
    END
    PARALLEL HOST LIBRARY:
        HOST DEFINITION: zhiciofcdell
            Remote Host Name = ZHICI-OFC-DELL
            Host Architecture String = winnt-amd64
            Installation Root = C:\Program Files\ANSYS Inc\v%v\CFX
        END
    END
    PARTITIONER STEP CONTROL:
        Multidomain Option = Independent Partitioning
        Runtime Priority = Standard
        EXECUTABLE SELECTION:
            Use Large Problem Partitioner = Off
        END

```

```

MEMORY CONTROL:
  Memory Allocation Factor = 1.0
END
PARTITIONING TYPE:
  MeTiS Type = k-way
  Option = MeTiS
  Partition Size Rule = Automatic
  Partition Weight Factors = 0.25000, 0.25000, 0.25000, 0.25000
END
END
RUN DEFINITION:
  Run Mode = Full
  Solver Input File = With BB 3D.def
END
SOLVER STEP CONTROL:
  Runtime Priority = Standard
  MEMORY CONTROL:
    Integer Memory Override = 1.3x
    Memory Allocation Factor = 1.0
    Real Memory Override = 1.5x
  END
  PARALLEL ENVIRONMENT:
    Number of Processes = 4
    Start Method = HP MPI Local Parallel
    Parallel Host List = zhiciofcdell*4
  END
END
END
END

```


Simulation settings for validating with Saha's experiment. (Section 5.3)

Setting up CFX Solver run ...

```
+-----+
|                                     |
|                                     CFX Command Language for Run          |
|                                     |
+-----+
```

LIBRARY:

CEL:

EXPRESSIONS:

```
Free Stream Wind Speed = 8.23[m s-1]
Rotating Speed = TSR[rad]*Free Stream Wind Speed /Turbine Radius
TSR = 0.5
Time Step = Time for 1 Rev /120
Time for 1 Rev = 2*pi[rad]/Rotating Speed
Time for 3 Revs = 3*Time for 1 Rev
Time for 6 Revs = 6*Time for 1 Rev
Time for 9 Revs = 9*Time for 1 Rev
Turbine Radius = 0.135 [m]
```

END

END

MATERIAL: Air at 25 C

```
Material Description = Air at 25 C and 1 atm (dry)
Material Group = Air Data, Constant Property Gases
Option = Pure Substance
Thermodynamic State = Gas
```

PROPERTIES:

Option = General Material

EQUATION OF STATE:

```
Density = 1.185 [kg m-3]
Molar Mass = 28.96 [kg kmol-1]
Option = Value
```

END

SPECIFIC HEAT CAPACITY:

```
Option = Value
Specific Heat Capacity = 1.0044E+03 [J kg-1 K-1]
Specific Heat Type = Constant Pressure
```

END

REFERENCE STATE:

```
Option = Specified Point
Reference Pressure = 1 [atm]
Reference Specific Enthalpy = 0. [J/kg]
Reference Specific Entropy = 0. [J/kg/K]
Reference Temperature = 25 [C]
```

END

DYNAMIC VISCOSITY:

```
Dynamic Viscosity = 1.831E-05 [kg m-1 s-1]
Option = Value
```

END

THERMAL CONDUCTIVITY:

```
Option = Value
Thermal Conductivity = 2.61E-02 [W m-1 K-1]
```

END

ABSORPTION COEFFICIENT:

```
Absorption Coefficient = 0.01 [m-1]
Option = Value
```

END

SCATTERING COEFFICIENT:

```
Option = Value
Scattering Coefficient = 0.0 [m-1]
```

END

REFRACTIVE INDEX:

```
Option = Value
Refractive Index = 1.0 [m m-1]
```

END

THERMAL EXPANSIVITY:

```
Option = Value
Thermal Expansivity = 0.003356 [K-1]
```

END

END

END

END

```

FLOW: Flow Analysis 1
SOLUTION UNITS:
  Angle Units = [rad]
  Length Units = [m]
  Mass Units = [kg]
  Solid Angle Units = [sr]
  Temperature Units = [K]
  Time Units = [s]
END
ANALYSIS TYPE:
  Option = Transient
EXTERNAL SOLVER COUPLING:
  Option = None
END
INITIAL TIME:
  Option = Automatic with Value
  Time = 0 [s]
END
TIME DURATION:
  Option = Total Time
  Total Time = Time for 9 Revs
END
TIME STEPS:
  Option = Timesteps
  Timesteps = Time Step
END
DOMAIN: Rotating Region
  Coord Frame = Coord 0
  Domain Type = Fluid
  Location = B994
BOUNDARY: Symmetry RR
  Boundary Type = SYMMETRY
  Location = Symmetry RR
END
BOUNDARY: Turbine Surfaces
  Boundary Type = WALL
  Frame Type = Rotating
  Location = Wall Turbine Surfaces
BOUNDARY CONDITIONS:
  MASS AND MOMENTUM:
    Option = No Slip Wall
  END
  WALL ROUGHNESS:
    Option = Smooth Wall
  END
END
BOUNDARY: Wake to RR Side 2
  Boundary Type = INTERFACE
  Location = Interface RR
BOUNDARY CONDITIONS:
  MASS AND MOMENTUM:
    Option = Conservative Interface Flux
  END
  TURBULENCE:
    Option = Conservative Interface Flux
  END
END
DOMAIN MODELS:
  BUOYANCY MODEL:
    Option = Non Buoyant
  END
  DOMAIN MOTION:
    Angular Velocity = -Rotating Speed
    Option = Rotating
  AXIS DEFINITION:
    Option = Coordinate Axis
    Rotation Axis = Coord 0.2
  END
END
MESH DEFORMATION:
  Option = None
END
REFERENCE PRESSURE:

```

```

        Reference Pressure = 1 [atm]
    END
END
FLUID DEFINITION: Fluid 1
    Material = Air at 25 C
    Option = Material Library
    MORPHOLOGY:
        Option = Continuous Fluid
    END
END
FLUID MODELS:
    COMBUSTION MODEL:
        Option = None
    END
    HEAT TRANSFER MODEL:
        Option = None
    END
    THERMAL RADIATION MODEL:
        Option = None
    END
    TURBULENCE MODEL:
        Option = SST
    END
    TURBULENT WALL FUNCTIONS:
        Option = Automatic
    END
END
INITIALISATION:
    Frame Type = Stationary
    Option = Automatic
    INITIAL CONDITIONS:
        Velocity Type = Cartesian
        CARTESIAN VELOCITY COMPONENTS:
            Option = Automatic with Value
            U = Free Stream Wind Speed
            V = 0 [m s-1]
            W = 0 [m s-1]
        END
        STATIC PRESSURE:
            Option = Automatic with Value
            Relative Pressure = 0 [Pa]
        END
        TURBULENCE INITIAL CONDITIONS:
            Option = Medium Intensity and Eddy Viscosity Ratio
        END
    END
END
DOMAIN: Surrounding Air
    Coord Frame = Coord 0
    Domain Type = Fluid
    Location = B995
    BOUNDARY: Inlet
        Boundary Type = INLET
        Location = Inlet
        BOUNDARY CONDITIONS:
            FLOW REGIME:
                Option = Subsonic
            END
            MASS AND MOMENTUM:
                Normal Speed = Free Stream Wind Speed
                Option = Normal Speed
            END
            TURBULENCE:
                Option = Medium Intensity and Eddy Viscosity Ratio
            END
        END
    END
END
BOUNDARY: Opening Front
    Boundary Type = OPENING
    Location = Opening Front
    BOUNDARY CONDITIONS:
        FLOW DIRECTION:
            Option = Normal to Boundary Condition
        END
        FLOW REGIME:

```

```

    Option = Subsonic
END
MASS AND MOMENTUM:
    Option = Opening Pressure and Direction
    Relative Pressure = 0 [Pa]
END
TURBULENCE:
    Option = Medium Intensity and Eddy Viscosity Ratio
END
END
BOUNDARY: Opening Outlet
    Boundary Type = OPENING
    Location = Opening Outlet
    BOUNDARY CONDITIONS:
        FLOW DIRECTION:
            Option = Normal to Boundary Condition
        END
        FLOW REGIME:
            Option = Subsonic
        END
        MASS AND MOMENTUM:
            Option = Opening Pressure and Direction
            Relative Pressure = 0 [Pa]
        END
        TURBULENCE:
            Option = Medium Intensity and Eddy Viscosity Ratio
        END
    END
END
BOUNDARY: SA to Wake Side 1
    Boundary Type = INTERFACE
    Location = Interface SA
    BOUNDARY CONDITIONS:
        MASS AND MOMENTUM:
            Option = Conservative Interface Flux
        END
        TURBULENCE:
            Option = Conservative Interface Flux
        END
    END
END
BOUNDARY: Symmetry SA
    Boundary Type = SYMMETRY
    Location = Symmetry SA
END
BOUNDARY: Symmetry Top Sides
    Boundary Type = SYMMETRY
    Location = Symmetry Top Sides
END
DOMAIN MODELS:
    BUOYANCY MODEL:
        Option = Non Buoyant
    END
    DOMAIN MOTION:
        Option = Stationary
    END
    MESH DEFORMATION:
        Option = None
    END
    REFERENCE PRESSURE:
        Reference Pressure = 1 [atm]
    END
END
FLUID DEFINITION: Fluid 1
    Material = Air at 25 C
    Option = Material Library
    MORPHOLOGY:
        Option = Continuous Fluid
    END
END
FLUID MODELS:
    COMBUSTION MODEL:
        Option = None
    END
    HEAT TRANSFER MODEL:

```

```

        Option = None
    END
    THERMAL RADIATION MODEL:
        Option = None
    END
    TURBULENCE MODEL:
        Option = SST
    END
    TURBULENT WALL FUNCTIONS:
        Option = Automatic
    END
END
INITIALISATION:
    Option = Automatic
    INITIAL CONDITIONS:
        Velocity Type = Cartesian
        CARTESIAN VELOCITY COMPONENTS:
            Option = Automatic with Value
            U = 0 [m s-1]
            V = 0 [m s-1]
            W = 0 [m s-1]
        END
        STATIC PRESSURE:
            Option = Automatic with Value
            Relative Pressure = 0 [Pa]
        END
        TURBULENCE INITIAL CONDITIONS:
            Option = Medium Intensity and Eddy Viscosity Ratio
        END
    END
END
DOMAIN: Wake Region
    Coord Frame = Coord 0
    Domain Type = Fluid
    Location = B993
    BOUNDARY: SA to Wake Side 2
        Boundary Type = INTERFACE
        Location = Interface Wake SA
        BOUNDARY CONDITIONS:
            MASS AND MOMENTUM:
                Option = Conservative Interface Flux
            END
            TURBULENCE:
                Option = Conservative Interface Flux
            END
        END
    END
    BOUNDARY: Symmetry Wake
        Boundary Type = SYMMETRY
        Location = Symmetry Wake
    END
    BOUNDARY: Wake to RR Side 1
        Boundary Type = INTERFACE
        Location = Interface Wake RR
        BOUNDARY CONDITIONS:
            MASS AND MOMENTUM:
                Option = Conservative Interface Flux
            END
            TURBULENCE:
                Option = Conservative Interface Flux
            END
        END
    END
    BOUNDARY: Wall Shaft End
        Boundary Type = WALL
        Location = Wall Shaft End
        BOUNDARY CONDITIONS:
            MASS AND MOMENTUM:
                Option = No Slip Wall
            END
            WALL ROUGHNESS:
                Option = Smooth Wall
            END
        END
    END
END

```

```

DOMAIN MODELS:
  BUOYANCY MODEL:
    Option = Non Buoyant
  END
  DOMAIN MOTION:
    Option = Stationary
  END
  MESH DEFORMATION:
    Option = None
  END
  REFERENCE PRESSURE:
    Reference Pressure = 1 [atm]
  END
END
FLUID DEFINITION: Fluid 1
  Material = Air at 25 C
  Option = Material Library
  MORPHOLOGY:
    Option = Continuous Fluid
  END
END
FLUID MODELS:
  COMBUSTION MODEL:
    Option = None
  END
  HEAT TRANSFER MODEL:
    Option = None
  END
  THERMAL RADIATION MODEL:
    Option = None
  END
  TURBULENCE MODEL:
    Option = SST
  END
  TURBULENT WALL FUNCTIONS:
    Option = Automatic
  END
END
INITIALISATION:
  Option = Automatic
  INITIAL CONDITIONS:
    Velocity Type = Cartesian
    CARTESIAN VELOCITY COMPONENTS:
      Option = Automatic with Value
      U = Free Stream Wind Speed
      V = 0 [m s-1]
      W = 0 [m s-1]
    END
    STATIC PRESSURE:
      Option = Automatic with Value
      Relative Pressure = 0 [Pa]
    END
    TURBULENCE INITIAL CONDITIONS:
      Option = Medium Intensity and Eddy Viscosity Ratio
    END
  END
END
DOMAIN INTERFACE: SA to Wake
  Boundary List1 = SA to Wake Side 1
  Boundary List2 = SA to Wake Side 2
  Interface Type = Fluid Fluid
  INTERFACE MODELS:
    Option = General Connection
  FRAME CHANGE:
    Option = None
  END
  MASS AND MOMENTUM:
    Option = Conservative Interface Flux
  MOMENTUM INTERFACE MODEL:
    Option = None
  END
END
PITCH CHANGE:
  Option = None
END

```

```

END
MESH CONNECTION:
  Option = GGI
END
END
DOMAIN INTERFACE: Wake to RR
  Boundary List1 = Wake to RR Side 1
  Boundary List2 = Wake to RR Side 2
  Interface Type = Fluid Fluid
  INTERFACE MODELS:
    Option = General Connection
    FRAME CHANGE:
      Option = Transient Rotor Stator
    END
    MASS AND MOMENTUM:
      Option = Conservative Interface Flux
    MOMENTUM INTERFACE MODEL:
      Option = None
    END
  END
  PITCH CHANGE:
    Option = None
  END
END
MESH CONNECTION:
  Option = GGI
END
END
OUTPUT CONTROL:
  RESULTS:
    File Compression Level = Default
    Option = Standard
  END
  TRANSIENT RESULTS: Transient Results 1
    File Compression Level = Default
    Include Mesh = No
    Option = Selected Variables
    Output Variables List = Pressure,Velocity in Stn Frame
    OUTPUT FREQUENCY:
      Option = Timestep Interval
      Timestep Interval = 10
    END
  END
  TRANSIENT RESULTS: Transient Results 2
    File Compression Level = Default
    Include Mesh = No
    Option = Selected Variables
    Output Variables List = Pressure,Velocity in Stn Frame
    OUTPUT FREQUENCY:
      Option = Timestep List
      Timestep List = 962, 964, 966, 968, 970, 972, 974, 976, 978, 980, \
        982, 984, 986, 988, 990, 992, 994, 996, 998, 1000, 1002, 1004, \
        1006, 1008, 1010, 1012, 1014, 1016, 1018, 1020, 1022, 1024, 1026, \
        1028, 1030, 1032, 1034, 1036, 1038, 1040, 1042, 1044, 1046, 1048, \
        1050, 1052, 1054, 1056, 1058, 1060, 1062, 1064, 1066, 1068, 1070, \
        1072, 1074, 1076, 1078, 1080
    END
  END
END
SOLVER CONTROL:
  Turbulence Numerics = First Order
  ADVECTION SCHEME:
    Option = High Resolution
  END
  CONVERGENCE CONTROL:
    Maximum Number of Coefficient Loops = 10
    Minimum Number of Coefficient Loops = 1
    Timescale Control = Coefficient Loops
  END
  CONVERGENCE CRITERIA:
    Residual Target = 0.001
    Residual Type = RMS
  END
  TRANSIENT SCHEME:
    Option = Second Order Backward Euler
  TIMESTEP INITIALISATION:

```

```

        Option = Automatic
    END
END
END
COMMAND FILE:
    Version = 13.0
    Results Version = 13.0
END
PARAMETERIZATION:
    INPUT FIELD: TSR
        Expression Name = TSR
        Method = Expression
    END
END
SIMULATION CONTROL:
    EXECUTION CONTROL:
        EXECUTABLE SELECTION:
            Double Precision = Off
        END
        INTERPOLATOR STEP CONTROL:
            Runtime Priority = Standard
        MEMORY CONTROL:
            Memory Allocation Factor = 1.0
        END
    END
    PARALLEL HOST LIBRARY:
        HOST DEFINITION: zhiciofcdell
            Remote Host Name = ZHICI-OFC-DELL
            Host Architecture String = winnt-amd64
            Installation Root = C:\Program Files\ANSYS Inc\v%v\CFX
        END
    END
    PARTITIONER STEP CONTROL:
        Multidomain Option = Independent Partitioning
        Runtime Priority = Standard
        EXECUTABLE SELECTION:
            Use Large Problem Partitioner = Off
        END
        MEMORY CONTROL:
            Memory Allocation Factor = 1.0
        END
        PARTITIONING TYPE:
            MeTiS Type = k-way
            Option = MeTiS
            Partition Size Rule = Automatic
            Partition Weight Factors = 0.25000, 0.25000, 0.25000, 0.25000
        END
    END
    RUN DEFINITION:
        Run Mode = Full
        Solver Input File = Saha Turbine.def
    END
    SOLVER STEP CONTROL:
        Runtime Priority = Standard
        MEMORY CONTROL:
            Integer Memory Override = 1.3x
            Memory Allocation Factor = 1.0
            Real Memory Override = 1.3x
        END
        PARALLEL ENVIRONMENT:
            Number of Processes = 4
            Start Method = HP MPI Local Parallel
            Parallel Host List = zhiciofcdell*4
        END
    END
END
END

```


Simulation settings for validating with driving test. (Section 5.4)

Setting up CFX Solver run ...

```
+-----+
|                                             |
|               CFX Command Language for Run   |
|                                             |
+-----+
```

LIBRARY:

CEL:

EXPRESSIONS:

```
Free Stream Wind Speed = 12.0809[m s-1]
Rotating Speed = TSR[rad]*Free Stream Wind Speed /Turbine Radius
TSR = 0.88306
Time Step = Time for 1 Rev /120
Time for 1 Rev = 2*pi[rad]/Rotating Speed
Time for 3 Revs = 3*Time for 1 Rev
Time for 6 Revs = 6*Time for 1 Rev
Time for 9 Revs = 9*Time for 1 Rev
Turbine Radius = 0.55[m]
```

END

END

MATERIAL: Air at 25 C

```
Material Description = Air at 25 C and 1 atm (dry)
Material Group = Air Data, Constant Property Gases
Option = Pure Substance
Thermodynamic State = Gas
```

PROPERTIES:

Option = General Material

EQUATION OF STATE:

```
Density = 1.185 [kg m-3]
Molar Mass = 28.96 [kg kmol-1]
Option = Value
```

END

SPECIFIC HEAT CAPACITY:

```
Option = Value
Specific Heat Capacity = 1.0044E+03 [J kg-1 K-1]
Specific Heat Type = Constant Pressure
```

END

REFERENCE STATE:

```
Option = Specified Point
Reference Pressure = 1 [atm]
Reference Specific Enthalpy = 0. [J/kg]
Reference Specific Entropy = 0. [J/kg/K]
Reference Temperature = 25 [C]
```

END

DYNAMIC VISCOSITY:

```
Dynamic Viscosity = 1.831E-05 [kg m-1 s-1]
Option = Value
```

END

THERMAL CONDUCTIVITY:

```
Option = Value
Thermal Conductivity = 2.61E-02 [W m-1 K-1]
```

END

ABSORPTION COEFFICIENT:

```
Absorption Coefficient = 0.01 [m-1]
Option = Value
```

END

SCATTERING COEFFICIENT:

```
Option = Value
Scattering Coefficient = 0.0 [m-1]
```

END

REFRACTIVE INDEX:

```
Option = Value
Refractive Index = 1.0 [m m-1]
```

END

THERMAL EXPANSIVITY:

```
Option = Value
Thermal Expansivity = 0.003356 [K-1]
```

END

END

END

END

```

FLOW: Flow Analysis 1
SOLUTION UNITS:
  Angle Units = [rad]
  Length Units = [m]
  Mass Units = [kg]
  Solid Angle Units = [sr]
  Temperature Units = [K]
  Time Units = [s]
END
ANALYSIS TYPE:
  Option = Transient
EXTERNAL SOLVER COUPLING:
  Option = None
END
INITIAL TIME:
  Option = Automatic with Value
  Time = 0 [s]
END
TIME DURATION:
  Option = Total Time
  Total Time = Time for 9 Revs
END
TIME STEPS:
  Option = Timesteps
  Timesteps = Time Step
END
DOMAIN: Rotating Region
  Coord Frame = Coord 0
  Domain Type = Fluid
  Location = Assembly 3
BOUNDARY: Domain Interface 1 Side 1 1
  Boundary Type = INTERFACE
  Location = Interface_RR
BOUNDARY CONDITIONS:
  MASS AND MOMENTUM:
    Option = Conservative Interface Flux
  END
  TURBULENCE:
    Option = Conservative Interface Flux
  END
END
BOUNDARY: Symmetry_RR
  Boundary Type = SYMMETRY
  Location = Symmetry_RR
END
BOUNDARY: Wall_Turbine_surfaces
  Boundary Type = WALL
  Frame Type = Rotating
  Location = Wall_turbine_surfaces
BOUNDARY CONDITIONS:
  MASS AND MOMENTUM:
    Option = No Slip Wall
  END
  WALL ROUGHNESS:
    Option = Smooth Wall
  END
END
END
DOMAIN MODELS:
  BUOYANCY MODEL:
    Option = Non Buoyant
  END
  DOMAIN MOTION:
    Angular Velocity = -Rotating Speed
    Option = Rotating
  AXIS DEFINITION:
    Option = Coordinate Axis
    Rotation Axis = Coord 0.3
  END
END
MESH DEFORMATION:
  Option = None
END
REFERENCE PRESSURE:

```

```

        Reference Pressure = 1 [atm]
    END
END
FLUID DEFINITION: Fluid 1
    Material = Air at 25 C
    Option = Material Library
    MORPHOLOGY:
        Option = Continuous Fluid
    END
END
FLUID MODELS:
    COMBUSTION MODEL:
        Option = None
    END
    HEAT TRANSFER MODEL:
        Option = None
    END
    THERMAL RADIATION MODEL:
        Option = None
    END
    TURBULENCE MODEL:
        Option = SST
    END
    TURBULENT WALL FUNCTIONS:
        Option = Automatic
    END
END
INITIALISATION:
    Frame Type = Stationary
    Option = Automatic
    INITIAL CONDITIONS:
        Velocity Type = Cartesian
        CARTESIAN VELOCITY COMPONENTS:
            Option = Automatic with Value
            U = Free Stream Wind Speed
            V = 0 [m s-1]
            W = 0 [m s-1]
        END
        STATIC PRESSURE:
            Option = Automatic with Value
            Relative Pressure = 0 [Pa]
        END
        TURBULENCE INITIAL CONDITIONS:
            Option = Medium Intensity and Eddy Viscosity Ratio
        END
    END
END
DOMAIN: Surrounding Air
    Coord Frame = Coord 0
    Domain Type = Fluid
    Location = Assembly 2
    BOUNDARY: Domain Interface 2 Side 1
        Boundary Type = INTERFACE
        Location = Interface_SA
        BOUNDARY CONDITIONS:
            MASS AND MOMENTUM:
                Option = Conservative Interface Flux
            END
            TURBULENCE:
                Option = Conservative Interface Flux
            END
        END
    END
END
BOUNDARY: Inlet
    Boundary Type = INLET
    Location = Inlet
    BOUNDARY CONDITIONS:
        FLOW REGIME:
            Option = Subsonic
        END
        MASS AND MOMENTUM:
            Normal Speed = Free Stream Wind Speed
            Option = Normal Speed
        END
        TURBULENCE:

```

```

        Option = Low Intensity and Eddy Viscosity Ratio
    END
END
BOUNDARY: Outlet
    Boundary Type = OUTLET
    Location = Outlet
    BOUNDARY CONDITIONS:
        FLOW REGIME:
            Option = Subsonic
        END
        MASS AND MOMENTUM:
            Option = Static Pressure
            Relative Pressure = 0 [Pa]
        END
    END
END
BOUNDARY: Symmetry_SA
    Boundary Type = SYMMETRY
    Location = Symmetry_SA
END
BOUNDARY: Symmetry_right
    Boundary Type = SYMMETRY
    Location = Symmetry_right
END
BOUNDARY: Wal_test_rigs
    Boundary Type = WALL
    Location = Wall_test_rig
    BOUNDARY CONDITIONS:
        MASS AND MOMENTUM:
            Option = No Slip Wall
        END
        WALL ROUGHNESS:
            Option = Smooth Wall
        END
    END
END
BOUNDARY: Wall_ground
    Boundary Type = WALL
    Location = Wall_ground
    BOUNDARY CONDITIONS:
        MASS AND MOMENTUM:
            Option = Free Slip Wall
        END
    END
END
DOMAIN MODELS:
    BUOYANCY MODEL:
        Option = Non Buoyant
    END
    DOMAIN MOTION:
        Option = Stationary
    END
    MESH DEFORMATION:
        Option = None
    END
    REFERENCE PRESSURE:
        Reference Pressure = 1 [atm]
    END
END
FLUID DEFINITION: Fluid 1
    Material = Air at 25 C
    Option = Material Library
    MORPHOLOGY:
        Option = Continuous Fluid
    END
END
FLUID MODELS:
    COMBUSTION MODEL:
        Option = None
    END
    HEAT TRANSFER MODEL:
        Option = None
    END
    THERMAL RADIATION MODEL:
        Option = None

```

```

END
TURBULENCE MODEL:
  Option = SST
END
TURBULENT WALL FUNCTIONS:
  Option = Automatic
END
END
INITIALISATION:
  Option = Automatic
  INITIAL CONDITIONS:
    Velocity Type = Cartesian
    CARTESIAN VELOCITY COMPONENTS:
      Option = Automatic with Value
      U = Free Stream Wind Speed
      V = 0 [m s-1]
      W = 0 [m s-1]
    END
    STATIC PRESSURE:
      Option = Automatic with Value
      Relative Pressure = 0 [Pa]
    END
    TURBULENCE INITIAL CONDITIONS:
      Option = Medium Intensity and Eddy Viscosity Ratio
    END
  END
END
DOMAIN: Wake Region
  Coord Frame = Coord 0
  Domain Type = Fluid
  Location = B1136
  BOUNDARY: Domain Interface 1 Side 2
    Boundary Type = INTERFACE
    Location = Interface_wake_RR
    BOUNDARY CONDITIONS:
      MASS AND MOMENTUM:
        Option = Conservative Interface Flux
      END
      TURBULENCE:
        Option = Conservative Interface Flux
      END
    END
  END
  BOUNDARY: Domain Interface 2 Side 2
    Boundary Type = INTERFACE
    Location = Interface_wake
    BOUNDARY CONDITIONS:
      MASS AND MOMENTUM:
        Option = Conservative Interface Flux
      END
      TURBULENCE:
        Option = Conservative Interface Flux
      END
    END
  END
  BOUNDARY: Symmetry_wake
    Boundary Type = SYMMETRY
    Location = Symmetry_wake
  END
  BOUNDARY: Wall_roof_top
    Boundary Type = WALL
    Location = Wall_roof_top
    BOUNDARY CONDITIONS:
      MASS AND MOMENTUM:
        Option = No Slip Wall
      END
      WALL ROUGHNESS:
        Option = Smooth Wall
      END
    END
  END
  BOUNDARY: Wall_turbine_support
    Boundary Type = WALL
    Location = Wall_turbine_support
    BOUNDARY CONDITIONS:

```

```

    MASS AND MOMENTUM:
        Option = No Slip Wall
    END
    WALL ROUGHNESS:
        Option = Smooth Wall
    END
END
DOMAIN MODELS:
    BUOYANCY MODEL:
        Option = Non Buoyant
    END
    DOMAIN MOTION:
        Option = Stationary
    END
    MESH DEFORMATION:
        Option = None
    END
    REFERENCE PRESSURE:
        Reference Pressure = 1 [atm]
    END
END
FLUID DEFINITION: Fluid 1
    Material = Air at 25 C
    Option = Material Library
    MORPHOLOGY:
        Option = Continuous Fluid
    END
END
FLUID MODELS:
    COMBUSTION MODEL:
        Option = None
    END
    HEAT TRANSFER MODEL:
        Option = None
    END
    THERMAL RADIATION MODEL:
        Option = None
    END
    TURBULENCE MODEL:
        Option = SST
    END
    TURBULENT WALL FUNCTIONS:
        Option = Automatic
    END
END
INITIALISATION:
    Option = Automatic
    INITIAL CONDITIONS:
        Velocity Type = Cartesian
        CARTESIAN VELOCITY COMPONENTS:
            Option = Automatic with Value
            U = Free Stream Wind Speed
            V = 0 [m s-1]
            W = 0 [m s-1]
        END
        STATIC PRESSURE:
            Option = Automatic with Value
            Relative Pressure = 0 [Pa]
        END
        TURBULENCE INITIAL CONDITIONS:
            Option = Medium Intensity and Eddy Viscosity Ratio
        END
    END
END
DOMAIN INTERFACE: Domain Interface 1
    Boundary List1 = Domain Interface 1 Side 1 1
    Boundary List2 = Domain Interface 1 Side 2
    Interface Type = Fluid Fluid
    INTERFACE MODELS:
        Option = General Connection
    FRAME CHANGE:
        Option = Transient Rotor Stator
    END
    MASS AND MOMENTUM:

```

```

        Option = Conservative Interface Flux
    MOMENTUM INTERFACE MODEL:
        Option = None
    END
END
PITCH CHANGE:
    Option = None
END
MESH CONNECTION:
    Option = GGI
END
END
DOMAIN INTERFACE: Domain Interface 2
    Boundary List1 = Domain Interface 2 Side 1
    Boundary List2 = Domain Interface 2 Side 2
    Interface Type = Fluid Fluid
    INTERFACE MODELS:
        Option = General Connection
    FRAME CHANGE:
        Option = None
    END
    MASS AND MOMENTUM:
        Option = Conservative Interface Flux
    MOMENTUM INTERFACE MODEL:
        Option = None
    END
    PITCH CHANGE:
        Option = None
    END
    MESH CONNECTION:
        Option = GGI
    END
END
OUTPUT CONTROL:
    RESULTS:
        File Compression Level = Default
        Option = Standard
    END
    TRANSIENT RESULTS: Transient Results 1
        File Compression Level = Default
        Include Mesh = No
        Option = Selected Variables
        Output Variables List = Velocity in Stn Frame,Velocity in Stn Frame \
            v,Velocity in Stn Frame u,Velocity in Stn Frame w,Pressure
        OUTPUT FREQUENCY:
            Option = Timestep Interval
            Timestep Interval = 10
        END
    END
    TRANSIENT RESULTS: Transient Results 2
        File Compression Level = Default
        Include Mesh = No
        Option = Selected Variables
        Output Variables List = Pressure,Velocity in Stn Frame,Velocity in Stn \
            Frame v,Velocity in Stn Frame w,Velocity in Stn Frame u
        OUTPUT FREQUENCY:
            Option = Timestep List
            Timestep List = 960, 962, 964, 966, 968, 970, 972, 974, 976, 978, \
                980, 982, 984, 986, 988, 990, 992, 994, 996, 998, 1000, 1002, \
                1004, 1006, 1008, 1010, 1012, 1014, 1016, 1018, 1020, 1022, 1024, \
                1026, 1028, 1030, 1032, 1034, 1036, 1038, 1040, 1042, 1044, 1046, \
                1048, 1050, 1052, 1054, 1056, 1058, 1060, 1062, 1064, 1066, 1068, \
                1070, 1072, 1074, 1076, 1078, 1080
        END
    END
END
SOLVER CONTROL:
    Turbulence Numerics = First Order
    ADVECTION SCHEME:
        Option = High Resolution
    END
    CONVERGENCE CONTROL:
        Maximum Number of Coefficient Loops = 10

```

```

        Minimum Number of Coefficient Loops = 1
        Timescale Control = Coefficient Loops
    END
    CONVERGENCE CRITERIA:
        Residual Target = 0.001
        Residual Type = RMS
    END
    TRANSIENT SCHEME:
        Option = Second Order Backward Euler
    TIMESTEP INITIALISATION:
        Option = Automatic
    END
END
END
END
COMMAND FILE:
    Version = 13.0
    Results Version = 13.0
END
PARAMETERIZATION:
    INPUT FIELD: TSR
        Expression Name = TSR
        Method = Expression
    END
END
SIMULATION CONTROL:
    EXECUTION CONTROL:
        EXECUTABLE SELECTION:
            Double Precision = Off
        END
        INTERPOLATOR STEP CONTROL:
            Runtime Priority = Standard
        MEMORY CONTROL:
            Memory Allocation Factor = 1.0
        END
    END
    PARALLEL HOST LIBRARY:
        HOST DEFINITION: dellprecision
            Remote Host Name = DELL-PRECISION
            Host Architecture String = winnt-amd64
            Installation Root = C:\Program Files\ANSYS Inc\v%v\CFX
        END
    END
    PARTITIONER STEP CONTROL:
        Multidomain Option = Independent Partitioning
        Runtime Priority = Standard
        EXECUTABLE SELECTION:
            Use Large Problem Partitioner = Off
        END
        MEMORY CONTROL:
            Memory Allocation Factor = 1.0
        END
        PARTITIONING TYPE:
            MeTiS Type = k-way
            Option = MeTiS
            Partition Size Rule = Automatic
            Partition Weight Factors = 0.12500, 0.12500, 0.12500, 0.12500, \
            0.12500, 0.12500, 0.12500, 0.12500
        END
    END
    RUN DEFINITION:
        Run Mode = Full
        Solver Input File = Fine Wake z10 y9.def
    END
    SOLVER STEP CONTROL:
        Runtime Priority = Standard
        MEMORY CONTROL:
            Integer Memory Override = 1.3x
            Memory Allocation Factor = 1.0
            Real Memory Override = 1.3x
        END
    PARALLEL ENVIRONMENT:
        Number of Processes = 8
        Start Method = HP MPI Local Parallel
        Parallel Host List = dellprecision*8
    END

```


END
END
END

Simulation settings for parametric study of a generic turbine above a bluff body. (Section 5.5.3)

Frozen rotor simulation:

Setting up CFX Solver run ...

```
+-----+
|                                             |
|               CFX Command Language for Run |
|                                             |
+-----+
```

LIBRARY:

CEL:

EXPRESSIONS:

FS Wind Speed Frozen Rotor = 12 [m s⁻¹]
Rotating Speed = TSR Frozen Rotor[rad]*FS Wind Speed Frozen \
Rotor/Turbine R Frozen Rotor
TSR Frozen Rotor = 1.3
Turbine R Frozen Rotor = 1.25 [m]

END

END

MATERIAL: Air at 25 C

Material Description = Air at 25 C and 1 atm (dry)
Material Group = Air Data, Constant Property Gases
Option = Pure Substance
Thermodynamic State = Gas

PROPERTIES:

Option = General Material

EQUATION OF STATE:

Density = 1.185 [kg m⁻³]
Molar Mass = 28.96 [kg kmol⁻¹]
Option = Value

END

SPECIFIC HEAT CAPACITY:

Option = Value
Specific Heat Capacity = 1.0044E+03 [J kg⁻¹ K⁻¹]
Specific Heat Type = Constant Pressure

END

REFERENCE STATE:

Option = Specified Point
Reference Pressure = 1 [atm]
Reference Specific Enthalpy = 0. [J/kg]
Reference Specific Entropy = 0. [J/kg/K]
Reference Temperature = 25 [C]

END

DYNAMIC VISCOSITY:

Dynamic Viscosity = 1.831E-05 [kg m⁻¹ s⁻¹]
Option = Value

END

THERMAL CONDUCTIVITY:

Option = Value
Thermal Conductivity = 2.61E-02 [W m⁻¹ K⁻¹]

END

ABSORPTION COEFFICIENT:

Absorption Coefficient = 0.01 [m⁻¹]
Option = Value

END

SCATTERING COEFFICIENT:

Option = Value
Scattering Coefficient = 0.0 [m⁻¹]

END

REFRACTIVE INDEX:

Option = Value
Refractive Index = 1.0 [m m⁻¹]

END

THERMAL EXPANSIVITY:

Option = Value
Thermal Expansivity = 0.003356 [K⁻¹]

END

END

```

END
END
FLOW: Flow Analysis 1
  SOLUTION UNITS:
    Angle Units = [rad]
    Length Units = [m]
    Mass Units = [kg]
    Solid Angle Units = [sr]
    Temperature Units = [K]
    Time Units = [s]
  END
  ANALYSIS TYPE:
    Option = Steady State
  EXTERNAL SOLVER COUPLING:
    Option = None
  END
END
DOMAIN: Rotating Region
  Coord Frame = Coord 0
  Domain Type = Fluid
  Location = B160
  BOUNDARY: Domain Interface 1 Side 1
    Boundary Type = INTERFACE
    Location = interface_RR
    BOUNDARY CONDITIONS:
      MASS AND MOMENTUM:
        Option = Conservative Interface Flux
      END
      TURBULENCE:
        Option = Conservative Interface Flux
      END
    END
  END
  BOUNDARY: Symmetry RR
    Boundary Type = SYMMETRY
    Location = symmetry_RR
  END
  BOUNDARY: Wall Turbine Surfaces
    Boundary Type = WALL
    Frame Type = Rotating
    Location = wall_turbine_surf
    BOUNDARY CONDITIONS:
      MASS AND MOMENTUM:
        Option = No Slip Wall
      END
      WALL ROUGHNESS:
        Option = Smooth Wall
      END
    END
  END
  DOMAIN MODELS:
    BUOYANCY MODEL:
      Option = Non Buoyant
    END
    DOMAIN MOTION:
      Angular Velocity = -Rotating Speed
      Option = Rotating
    AXIS DEFINITION:
      Option = Coordinate Axis
      Rotation Axis = Coord 0.3
    END
  END
  MESH DEFORMATION:
    Option = None
  END
  REFERENCE PRESSURE:
    Reference Pressure = 1 [atm]
  END
END
FLUID DEFINITION: Fluid 1
  Material = Air at 25 C
  Option = Material Library
  MORPHOLOGY:
    Option = Continuous Fluid
  END
END

```

```

FLUID MODELS:
  COMBUSTION MODEL:
    Option = None
  END
  HEAT TRANSFER MODEL:
    Option = None
  END
  THERMAL RADIATION MODEL:
    Option = None
  END
  TURBULENCE MODEL:
    Option = SST
  END
  TURBULENT WALL FUNCTIONS:
    Option = Automatic
  END
END
INITIALISATION:
  Frame Type = Stationary
  Option = Automatic
  INITIAL CONDITIONS:
    Velocity Type = Cartesian
    CARTESIAN VELOCITY COMPONENTS:
      Option = Automatic with Value
      U = FS Wind Speed Frozen Rotor
      V = 0 [m s-1]
      W = 0 [m s-1]
    END
    STATIC PRESSURE:
      Option = Automatic with Value
      Relative Pressure = 0 [Pa]
    END
    TURBULENCE INITIAL CONDITIONS:
      Option = Medium Intensity and Eddy Viscosity Ratio
    END
  END
END
END
DOMAIN: Surrounding Air
  Coord Frame = Coord 0
  Domain Type = Fluid
  Location = Assembly
  BOUNDARY: Domain Interface 2 Side 2
    Boundary Type = INTERFACE
    Location = interface_SA
    BOUNDARY CONDITIONS:
      MASS AND MOMENTUM:
        Option = Conservative Interface Flux
      END
      TURBULENCE:
        Option = Conservative Interface Flux
      END
    END
  END
END
BOUNDARY: Inlet
  Boundary Type = INLET
  Location = inlet
  BOUNDARY CONDITIONS:
    FLOW REGIME:
      Option = Subsonic
    END
    MASS AND MOMENTUM:
      Normal Speed = FS Wind Speed Frozen Rotor
      Option = Normal Speed
    END
    TURBULENCE:
      Option = Medium Intensity and Eddy Viscosity Ratio
    END
  END
END
BOUNDARY: Outlet
  Boundary Type = OUTLET
  Location = outlet
  BOUNDARY CONDITIONS:
    FLOW REGIME:
      Option = Subsonic

```

```

END
MASS AND MOMENTUM:
  Option = Static Pressure
  Relative Pressure = 0 [Pa]
END
END
BOUNDARY: Symmetry SA
  Boundary Type = SYMMETRY
  Location = symmetry_SA
END
BOUNDARY: Symmetry Top Chamfer Side
  Boundary Type = SYMMETRY
  Location = symmetry_topchamferside
END
BOUNDARY: Wall SA
  Boundary Type = WALL
  Location = wall_SA_near_step,wall_others
BOUNDARY CONDITIONS:
  MASS AND MOMENTUM:
    Option = No Slip Wall
  END
  WALL ROUGHNESS:
    Option = Smooth Wall
  END
END
DOMAIN MODELS:
  BUOYANCY MODEL:
    Option = Non Buoyant
  END
  DOMAIN MOTION:
    Option = Stationary
  END
  MESH DEFORMATION:
    Option = None
  END
  REFERENCE PRESSURE:
    Reference Pressure = 1 [atm]
  END
END
FLUID DEFINITION: Fluid 1
  Material = Air at 25 C
  Option = Material Library
MORPHOLOGY:
  Option = Continuous Fluid
END
END
FLUID MODELS:
  COMBUSTION MODEL:
    Option = None
  END
  HEAT TRANSFER MODEL:
    Option = None
  END
  THERMAL RADIATION MODEL:
    Option = None
  END
  TURBULENCE MODEL:
    Option = SST
  END
  TURBULENT WALL FUNCTIONS:
    Option = Automatic
  END
END
INITIALISATION:
  Option = Automatic
  INITIAL CONDITIONS:
    Velocity Type = Cartesian
    CARTESIAN VELOCITY COMPONENTS:
      Option = Automatic with Value
      U = FS Wind Speed Frozen Rotor
      V = 0 [m s-1]
      W = 0 [m s-1]
    END
  STATIC PRESSURE:

```

```

        Option = Automatic with Value
        Relative Pressure = 0 [Pa]
    END
    TURBULENCE INITIAL CONDITIONS:
        Option = Medium Intensity and Eddy Viscosity Ratio
    END
END
END
DOMAIN: Wake Region
    Coord Frame = Coord 0
    Domain Type = Fluid
    Location = Assembly 2
    BOUNDARY: Domain Interface 1 Side 2
        Boundary Type = INTERFACE
        Location = interface_RRWR
        BOUNDARY CONDITIONS:
            MASS AND MOMENTUM:
                Option = Conservative Interface Flux
            END
            TURBULENCE:
                Option = Conservative Interface Flux
            END
        END
    END
    BOUNDARY: Domain Interface 2 Side 1
        Boundary Type = INTERFACE
        Location = interface_WRSA
        BOUNDARY CONDITIONS:
            MASS AND MOMENTUM:
                Option = Conservative Interface Flux
            END
            TURBULENCE:
                Option = Conservative Interface Flux
            END
        END
    END
    BOUNDARY: Symmetry WR
        Boundary Type = SYMMETRY
        Location = symmetry_WR
    END
    BOUNDARY: Wall WR
        Boundary Type = WALL
        Location = wall_WR_floor
        BOUNDARY CONDITIONS:
            MASS AND MOMENTUM:
                Option = No Slip Wall
            END
            WALL ROUGHNESS:
                Option = Smooth Wall
            END
        END
    END
    DOMAIN MODELS:
        BUOYANCY MODEL:
            Option = Non Buoyant
        END
        DOMAIN MOTION:
            Option = Stationary
        END
        MESH DEFORMATION:
            Option = None
        END
        REFERENCE PRESSURE:
            Reference Pressure = 1 [atm]
        END
    END
    FLUID DEFINITION: Fluid 1
        Material = Air at 25 C
        Option = Material Library
    MORPHOLOGY:
        Option = Continuous Fluid
    END
END
    FLUID MODELS:
        COMBUSTION MODEL:

```

```

        Option = None
    END
    HEAT TRANSFER MODEL:
        Option = None
    END
    THERMAL RADIATION MODEL:
        Option = None
    END
    TURBULENCE MODEL:
        Option = SST
    END
    TURBULENT WALL FUNCTIONS:
        Option = Automatic
    END
END
INITIALISATION:
    Option = Automatic
    INITIAL CONDITIONS:
        Velocity Type = Cartesian
        CARTESIAN VELOCITY COMPONENTS:
            Option = Automatic with Value
            U = FS Wind Speed Frozen Rotor
            V = 0 [m s-1]
            W = 0 [m s-1]
        END
        STATIC PRESSURE:
            Option = Automatic with Value
            Relative Pressure = 0 [Pa]
        END
        TURBULENCE INITIAL CONDITIONS:
            Option = Medium Intensity and Eddy Viscosity Ratio
        END
    END
END
END
DOMAIN INTERFACE: Domain Interface 1
    Boundary List1 = Domain Interface 1 Side 1
    Boundary List2 = Domain Interface 1 Side 2
    Interface Type = Fluid Fluid
    INTERFACE MODELS:
        Option = General Connection
    FRAME CHANGE:
        Option = Frozen Rotor
    END
    MASS AND MOMENTUM:
        Option = Conservative Interface Flux
    MOMENTUM INTERFACE MODEL:
        Option = None
    END
END
    PITCH CHANGE:
        Option = None
    END
END
    MESH CONNECTION:
        Option = GGI
    END
END
DOMAIN INTERFACE: Domain Interface 2
    Boundary List1 = Domain Interface 2 Side 1
    Boundary List2 = Domain Interface 2 Side 2
    Interface Type = Fluid Fluid
    INTERFACE MODELS:
        Option = General Connection
    FRAME CHANGE:
        Option = None
    END
    MASS AND MOMENTUM:
        Option = Conservative Interface Flux
    MOMENTUM INTERFACE MODEL:
        Option = None
    END
END
    PITCH CHANGE:
        Option = None
    END
END

```

```

END
MESH CONNECTION:
  Option = GGI
END
END
OUTPUT CONTROL:
  MONITOR OBJECTS:
    MONITOR BALANCES:
      Option = Full
    END
    MONITOR FORCES:
      Option = Full
    END
    MONITOR PARTICLES:
      Option = Full
    END
    MONITOR POINT: Outlet Ave Vel
      Expression Value = areaAve(Velocity in Stn Frame )@Outlet
      Option = Expression
    END
    MONITOR RESIDUALS:
      Option = Full
    END
    MONITOR TOTALS:
      Option = Full
    END
  END
RESULTS:
  File Compression Level = Default
  Option = Standard
END
END
SOLVER CONTROL:
  Turbulence Numerics = First Order
ADVECTION SCHEME:
  Option = High Resolution
END
CONVERGENCE CONTROL:
  Length Scale Option = Conservative
  Maximum Number of Iterations = 200
  Minimum Number of Iterations = 1
  Timescale Control = Auto Timescale
  Timescale Factor = 1.0
END
CONVERGENCE CRITERIA:
  Residual Target = 0.0001
  Residual Type = RMS
END
DYNAMIC MODEL CONTROL:
  Global Dynamic Model Control = Off
END
END
END
COMMAND FILE:
  Version = 13.0
  Results Version = 13.0
END
PARAMETERIZATION:
  INPUT FIELD: FS Wind Speed Frozen Rotor
    Expression Name = FS Wind Speed Frozen Rotor
    Method = Expression
  END
  INPUT FIELD: TSR Frozen Rotor
    Expression Name = TSR Frozen Rotor
    Method = Expression
  END
  INPUT FIELD: Turbine R Frozen Rotor
    Expression Name = Turbine R Frozen Rotor
    Method = Expression
  END
END
SIMULATION CONTROL:
  EXECUTION CONTROL:
    EXECUTABLE SELECTION:
      Double Precision = Off
    END
  END

```



```

INTERPOLATOR STEP CONTROL:
  Runtime Priority = Standard
MEMORY CONTROL:
  Memory Allocation Factor = 1.0
END
END
PARALLEL HOST LIBRARY:
  HOST DEFINITION: zhiciofcdell
    Remote Host Name = ZHICI-OFC-DELL
    Host Architecture String = winnt-amd64
    Installation Root = C:\Program Files\ANSYS Inc\v%v\CFX
  END
END
PARTITIONER STEP CONTROL:
  Multidomain Option = Independent Partitioning
  Runtime Priority = Standard
EXECUTABLE SELECTION:
  Use Large Problem Partitioner = Off
END
MEMORY CONTROL:
  Memory Allocation Factor = 1.0
END
PARTITIONING TYPE:
  MeTiS Type = k-way
  Option = MeTiS
  Partition Size Rule = Automatic
  Partition Weight Factors = 0.25000, 0.25000, 0.25000, 0.25000
END
END
RUN DEFINITION:
  Run Mode = Full
  Solver Input File = Frozen Rotor.def
END
SOLVER STEP CONTROL:
  Runtime Priority = Standard
MEMORY CONTROL:
  Memory Allocation Factor = 1.0
END
PARALLEL ENVIRONMENT:
  Number of Processes = 4
  Start Method = HP MPI Local Parallel
  Parallel Host List = zhiciofcdell*4
END
END
END
END

```

Transient simulation:

Setting up CFX Solver run ...

```

+-----+
|                                     |
|                                     CFX Command Language for Run          |
|                                     |
+-----+

```

```

LIBRARY:
CEL:
  EXPRESSIONS:
    FS Wind Speed Transient = 12 [m s^-1]
    Rotating Speed = TSR Transient[rad]*FS Wind Speed Transient/Turbine R \
      Transient
    TSR Transient = 1.3
    Time Step = Time for 1 Rev /120
    Time for 1 Rev = 2*pi[rad]/Rotating Speed
    Time for 2pt5 Rev = 2.5*Time for 1 Rev
    Turbine R Transient = 1.25 [m]
  END
END
MATERIAL: Air at 25 C
  Material Description = Air at 25 C and 1 atm (dry)
  Material Group = Air Data, Constant Property Gases
  Option = Pure Substance

```

```

Thermodynamic State = Gas
PROPERTIES:
  Option = General Material
  EQUATION OF STATE:
    Density = 1.185 [kg m^-3]
    Molar Mass = 28.96 [kg kmol^-1]
    Option = Value
  END
  SPECIFIC HEAT CAPACITY:
    Option = Value
    Specific Heat Capacity = 1.0044E+03 [J kg^-1 K^-1]
    Specific Heat Type = Constant Pressure
  END
  REFERENCE STATE:
    Option = Specified Point
    Reference Pressure = 1 [atm]
    Reference Specific Enthalpy = 0. [J/kg]
    Reference Specific Entropy = 0. [J/kg/K]
    Reference Temperature = 25 [C]
  END
  DYNAMIC VISCOSITY:
    Dynamic Viscosity = 1.831E-05 [kg m^-1 s^-1]
    Option = Value
  END
  THERMAL CONDUCTIVITY:
    Option = Value
    Thermal Conductivity = 2.61E-02 [W m^-1 K^-1]
  END
  ABSORPTION COEFFICIENT:
    Absorption Coefficient = 0.01 [m^-1]
    Option = Value
  END
  SCATTERING COEFFICIENT:
    Option = Value
    Scattering Coefficient = 0.0 [m^-1]
  END
  REFRACTIVE INDEX:
    Option = Value
    Refractive Index = 1.0 [m m^-1]
  END
  THERMAL EXPANSIVITY:
    Option = Value
    Thermal Expansivity = 0.003356 [K^-1]
  END
END
END
END
FLOW: Flow Analysis 1
  SOLUTION UNITS:
    Angle Units = [rad]
    Length Units = [m]
    Mass Units = [kg]
    Solid Angle Units = [sr]
    Temperature Units = [K]
    Time Units = [s]
  END
  ANALYSIS TYPE:
    Option = Transient
  EXTERNAL SOLVER COUPLING:
    Option = None
  END
  INITIAL TIME:
    Option = Automatic with Value
    Time = 0 [s]
  END
  TIME DURATION:
    Option = Time per run
    Time per run = Time for 2pt5 Rev
  END
  TIME STEPS:
    Option = Timesteps
    Timesteps = Time Step
  END
END
DOMAIN: Rotating Region
  Coord Frame = Coord 0

```

```

Domain Type = Fluid
Location = B160
BOUNDARY: Domain Interface 1 Side 1
  Boundary Type = INTERFACE
  Location = interface_RR
  BOUNDARY CONDITIONS:
    MASS AND MOMENTUM:
      Option = Conservative Interface Flux
    END
    TURBULENCE:
      Option = Conservative Interface Flux
    END
  END
END
BOUNDARY: Symmetry RR
  Boundary Type = SYMMETRY
  Location = symmetry_RR
END
BOUNDARY: Wall Turbine Surfaces
  Boundary Type = WALL
  Frame Type = Rotating
  Location = wall_turbine_surf
  BOUNDARY CONDITIONS:
    MASS AND MOMENTUM:
      Option = No Slip Wall
    END
    WALL ROUGHNESS:
      Option = Smooth Wall
    END
  END
END
DOMAIN MODELS:
  BUOYANCY MODEL:
    Option = Non Buoyant
  END
  DOMAIN MOTION:
    Angular Velocity = -Rotating Speed
    Option = Rotating
  AXIS DEFINITION:
    Option = Coordinate Axis
    Rotation Axis = Coord 0.3
  END
END
MESH DEFORMATION:
  Option = None
END
REFERENCE PRESSURE:
  Reference Pressure = 1 [atm]
END
END
FLUID DEFINITION: Fluid 1
  Material = Air at 25 C
  Option = Material Library
  MORPHOLOGY:
    Option = Continuous Fluid
  END
END
FLUID MODELS:
  COMBUSTION MODEL:
    Option = None
  END
  HEAT TRANSFER MODEL:
    Option = None
  END
  THERMAL RADIATION MODEL:
    Option = None
  END
  TURBULENCE MODEL:
    Option = SST
  END
  TURBULENT WALL FUNCTIONS:
    Option = Automatic
  END
END
END
DOMAIN: Surrounding Air

```

```

Coord Frame = Coord 0
Domain Type = Fluid
Location = Assembly
BOUNDARY: Domain Interface 2 Side 2
  Boundary Type = INTERFACE
  Location = interface_SA
  BOUNDARY CONDITIONS:
    MASS AND MOMENTUM:
      Option = Conservative Interface Flux
    END
    TURBULENCE:
      Option = Conservative Interface Flux
    END
  END
END
BOUNDARY: Inlet
  Boundary Type = INLET
  Location = inlet
  BOUNDARY CONDITIONS:
    FLOW REGIME:
      Option = Subsonic
    END
    MASS AND MOMENTUM:
      Normal Speed = FS Wind Speed Transient
      Option = Normal Speed
    END
    TURBULENCE:
      Option = Medium Intensity and Eddy Viscosity Ratio
    END
  END
END
BOUNDARY: Outlet
  Boundary Type = OUTLET
  Location = outlet
  BOUNDARY CONDITIONS:
    FLOW REGIME:
      Option = Subsonic
    END
    MASS AND MOMENTUM:
      Option = Static Pressure
      Relative Pressure = 0 [Pa]
    END
  END
END
BOUNDARY: Symmetry SA
  Boundary Type = SYMMETRY
  Location = symmetry_SA
END
BOUNDARY: Symmetry Top Chamfer Side
  Boundary Type = SYMMETRY
  Location = symmetry_topchamferside
END
BOUNDARY: Wall SA
  Boundary Type = WALL
  Location = wall_SA_near_step,wall_others
  BOUNDARY CONDITIONS:
    MASS AND MOMENTUM:
      Option = No Slip Wall
    END
    WALL ROUGHNESS:
      Option = Smooth Wall
    END
  END
END
DOMAIN MODELS:
  BUOYANCY MODEL:
    Option = Non Buoyant
  END
  DOMAIN MOTION:
    Option = Stationary
  END
  MESH DEFORMATION:
    Option = None
  END
  REFERENCE PRESSURE:
    Reference Pressure = 1 [atm]

```

```

END
END
FLUID DEFINITION: Fluid 1
  Material = Air at 25 C
  Option = Material Library
  MORPHOLOGY:
    Option = Continuous Fluid
  END
END
FLUID MODELS:
  COMBUSTION MODEL:
    Option = None
  END
  HEAT TRANSFER MODEL:
    Option = None
  END
  THERMAL RADIATION MODEL:
    Option = None
  END
  TURBULENCE MODEL:
    Option = SST
  END
  TURBULENT WALL FUNCTIONS:
    Option = Automatic
  END
END
END
DOMAIN: Wake Region
  Coord Frame = Coord 0
  Domain Type = Fluid
  Location = Assembly 2
  BOUNDARY: Domain Interface 1 Side 2
    Boundary Type = INTERFACE
    Location = interface_RRWR
    BOUNDARY CONDITIONS:
      MASS AND MOMENTUM:
        Option = Conservative Interface Flux
      END
      TURBULENCE:
        Option = Conservative Interface Flux
      END
    END
  END
  BOUNDARY: Domain Interface 2 Side 1
    Boundary Type = INTERFACE
    Location = interface_WRSA
    BOUNDARY CONDITIONS:
      MASS AND MOMENTUM:
        Option = Conservative Interface Flux
      END
      TURBULENCE:
        Option = Conservative Interface Flux
      END
    END
  END
  BOUNDARY: Symmetry WR
    Boundary Type = SYMMETRY
    Location = symmetry_WR
  END
  BOUNDARY: Wall WR
    Boundary Type = WALL
    Location = wall_WR_floor
    BOUNDARY CONDITIONS:
      MASS AND MOMENTUM:
        Option = No Slip Wall
      END
      WALL ROUGHNESS:
        Option = Smooth Wall
      END
    END
  END
DOMAIN MODELS:
  BUOYANCY MODEL:
    Option = Non Buoyant
  END
  DOMAIN MOTION:

```

```

        Option = Stationary
    END
    MESH DEFORMATION:
        Option = None
    END
    REFERENCE PRESSURE:
        Reference Pressure = 1 [atm]
    END
END
FLUID DEFINITION: Fluid 1
    Material = Air at 25 C
    Option = Material Library
    MORPHOLOGY:
        Option = Continuous Fluid
    END
END
FLUID MODELS:
    COMBUSTION MODEL:
        Option = None
    END
    HEAT TRANSFER MODEL:
        Option = None
    END
    THERMAL RADIATION MODEL:
        Option = None
    END
    TURBULENCE MODEL:
        Option = SST
    END
    TURBULENT WALL FUNCTIONS:
        Option = Automatic
    END
END
END
DOMAIN INTERFACE: Domain Interface 1
    Boundary List1 = Domain Interface 1 Side 1
    Boundary List2 = Domain Interface 1 Side 2
    Interface Type = Fluid Fluid
    INTERFACE MODELS:
        Option = General Connection
    FRAME CHANGE:
        Option = Transient Rotor Stator
    END
    MASS AND MOMENTUM:
        Option = Conservative Interface Flux
    MOMENTUM INTERFACE MODEL:
        Option = None
    END
    END
    PITCH CHANGE:
        Option = None
    END
END
MESH CONNECTION:
    Option = GGI
END
END
DOMAIN INTERFACE: Domain Interface 2
    Boundary List1 = Domain Interface 2 Side 1
    Boundary List2 = Domain Interface 2 Side 2
    Interface Type = Fluid Fluid
    INTERFACE MODELS:
        Option = General Connection
    FRAME CHANGE:
        Option = None
    END
    MASS AND MOMENTUM:
        Option = Conservative Interface Flux
    MOMENTUM INTERFACE MODEL:
        Option = None
    END
    END
    PITCH CHANGE:
        Option = None
    END
END
END

```

```

MESH CONNECTION:
  Option = GGI
END
END
OUTPUT CONTROL:
  MONITOR OBJECTS:
    MONITOR BALANCES:
      Option = Full
    END
    MONITOR FORCES:
      Option = Full
    END
    MONITOR PARTICLES:
      Option = Full
    END
    MONITOR POINT: Outlet Ave Vel
      Expression Value = areaAve(Velocity in Stn Frame )@Outlet
      Option = Expression
    END
    MONITOR RESIDUALS:
      Option = Full
    END
    MONITOR TOTALS:
      Option = Full
    END
  END
RESULTS:
  File Compression Level = Default
  Option = Standard
END
TRANSIENT RESULTS: Transient Results 1
  File Compression Level = Default
  Include Mesh = No
  Option = Selected Variables
  Output Variables List = Pressure,Velocity in Stn Frame,Velocity in Stn \
    Frame u,Velocity in Stn Frame v,Velocity in Stn Frame w
  OUTPUT FREQUENCY:
    Option = Timestep Interval
    Timestep Interval = 2
  END
END
SOLVER CONTROL:
  Turbulence Numerics = First Order
  ADVECTION SCHEME:
    Option = High Resolution
  END
  CONVERGENCE CONTROL:
    Maximum Number of Coefficient Loops = 10
    Minimum Number of Coefficient Loops = 1
    Timescale Control = Coefficient Loops
  END
  CONVERGENCE CRITERIA:
    Residual Target = 0.0001
    Residual Type = RMS
  END
  TRANSIENT SCHEME:
    Option = Second Order Backward Euler
  TIMESTEP INITIALISATION:
    Option = Automatic
  END
END
END
COMMAND FILE:
  Version = 13.0
  Results Version = 13.0
END
PARAMETERIZATION:
  INPUT FIELD: FS Wind Speed Transient
    Expression Name = FS Wind Speed Transient
    Method = Expression
  END
  INPUT FIELD: TSR Transient
    Expression Name = TSR Transient
    Method = Expression

```

```

END
INPUT FIELD: Turbine R Transient
  Expression Name = Turbine R Transient
  Method = Expression
END
END
SIMULATION CONTROL:
EXECUTION CONTROL:
  EXECUTABLE SELECTION:
    Double Precision = Off
  END
  INTERPOLATOR STEP CONTROL:
    Runtime Priority = Standard
  MEMORY CONTROL:
    Memory Allocation Factor = 1.0
  END
END
PARALLEL HOST LIBRARY:
  HOST DEFINITION: zhiciofcdell
    Remote Host Name = ZHICI-OFC-DELL
    Host Architecture String = winnt-amd64
    Installation Root = C:\Program Files\ANSYS Inc\v%v\CFX
  END
END
PARTITIONER STEP CONTROL:
  Multidomain Option = Independent Partitioning
  Runtime Priority = Standard
  EXECUTABLE SELECTION:
    Use Large Problem Partitioner = Off
  END
  MEMORY CONTROL:
    Memory Allocation Factor = 1.0
  END
  PARTITIONING TYPE:
    MeTiS Type = k-way
    Option = MeTiS
    Partition Size Rule = Automatic
    Partition Weight Factors = 0.25000, 0.25000, 0.25000, 0.25000
  END
END
RUN DEFINITION:
  Run Mode = Full
  Solver Input File = Transient.def
  INITIAL VALUES SPECIFICATION:
    INITIAL VALUES CONTROL:
      Use Mesh From = Solver Input File
      Continue History From = Workbench Initial Values
    END
    INITIAL VALUES: Workbench Initial Values
      Option = Results File
      File Name = Frozen Rotor_003.res
    END
  END
END
SOLVER STEP CONTROL:
  Runtime Priority = Standard
  MEMORY CONTROL:
    Integer Memory Override = 1.3x
    Memory Allocation Factor = 1.0
    Real Memory Override = 1.3x
  END
  PARALLEL ENVIRONMENT:
    Number of Processes = 4
    Start Method = HP MPI Local Parallel
    Parallel Host List = zhiciofcdell*4
  END
END
END
END

```