

Hydrodynamic Analysis of a Tidal Cross-Flow Turbine



A thesis submitted in partial fulfilment of the requirements
for the degree of Doctor of Philosophy.

Claudio A Consul

Worcester College

DPhil, Trinity 2011

Hydrodynamic Analysis of a Tidal Cross-Flow Turbine

Claudio A Consul, Worcester College. DPhil, Trinity 2011

Abstract

This study presents a numerical investigation of a generic horizontal axis cross-flow marine turbine. The numerical tool used is the commercial Computational Fluid Dynamics package ANSYS FLUENT 12.0. The numerical model, using the SST $k - \omega$ turbulence model, is validated against static, dynamic pitching blade and rotating turbine data.

The work embodies two main investigations. The first is concerned with the influence of turbine solidity (ratio of net blade chord to circumference) on turbine performance, and the second with the influence of blockage (ratio of device frontal area to channel cross-section area) and free surface deformation on the hydrodynamics of energy extraction in a constrained channel.

Turbine solidity was investigated by simulating flows through two-, three- and four-bladed turbines, resulting in solidities of 0.019, 0.029 and 0.038, respectively. The investigation was conducted for two Reynolds numbers, $Re = O(10^5)$ & $O(10^6)$, to reflect laboratory and field scales. Increasing the number of blades from two to four led to an increase in the maximum power coefficient from 0.43 to 0.53 for the lower Re and from 0.49 to 0.56 for the higher Re computations. Furthermore, the power curve was found to shift to a lower range of tip speed ratios when increasing solidity.

The effects of flow confinement and free surface deformation were investigated by simulating flows through a three-bladed turbine with solidity 0.125 at $Re = O(10^6)$ for channels that resulted in cross-stream blockages of 12.5% to 50%. Increasing the blockage led to a substantial increase in the power and basin efficiency; when approximating the free surface as a rigid lid, the highest power coefficient and basin efficiency computed were 1.18 and 0.54, respectively. Comparisons between the corresponding rigid

lid and free surface simulations, where Froude number, $Fr = 0.082$, rendered similar results at the lower blockages, but at the highest blockage an increase in power and basin efficiency of up to 7% for the free surface simulations over that achieved with a rigid lid boundary condition. For the free surface simulations with $Fr = 0.082$, the energy extraction resulted in a drop in water depth of up to 0.7%. An increase in Fr from 0.082 to 0.131 resulted in an increase maximum power of 3%, but a drop in basin efficiency of 21%.

Contents

1	Introduction	1
1.1	Outline of thesis	1
1.2	Renewable & tidal energy	3
1.2.1	Tidal dynamics	4
1.2.2	Tidal resources	5
1.2.3	Potential sites for tidal stream energy generation in the UK . .	6
1.2.4	Cost of tidal stream energy	9
1.3	Overview of technological status	11
1.3.1	Axial-flow turbines	11
1.3.1.1	Unducted turbine designs	11
1.3.1.2	Momentum actuator disc concept	14
1.3.1.3	Blade element theory	18
1.3.1.4	Ducted turbine designs	20
1.3.1.5	Turbine solidity	23
1.3.2	Cross-flow turbines	25
1.3.2.1	Transverse Horizontal Axis Water Turbine (THAWT) .	28
1.4	Summary of research on cross-flow turbines	30

2	Numerical Methods	35
2.1	Modelling techniques	36
2.1.1	Blade element momentum theory	36
2.1.2	Vortex models	38
2.1.3	Computational Fluid Dynamics (CFD)	39
2.2	Present numerical model - ANSYS FLUENT 12.0	43
2.2.1	Turbulence - RANS equations	43
2.2.2	Turbulence models	45
2.2.2.1	The Boussinesq approximation	45
2.2.2.2	Zero-equation models	46
2.2.2.3	One-equation models	46
2.2.2.4	Two-equation models	47
2.2.2.5	Spalart-Allmaras (S-A)	48
2.2.2.6	Shear Stress Transport (SST) $k - \omega$	50
2.2.3	Spatial discretisation: Finite Volume Method	52
2.2.4	Temporal discretisation	56
2.2.5	Free surface model - Volume of Fluid (VOF)	58
3	Validation	62
3.1	Reynolds number = $O(10^4)$ - $O(10^5)$	63
3.1.1	Static blade tests	63
3.1.1.1	Comparison of numerically and experimentally obtained lift and drag data	63

3.1.1.2	Spatial convergence & turbulence model tests	67
3.1.2	Rotating turbine tests	74
3.2	Reynolds number = $O(10^6)$	79
3.2.1	Static blade tests	79
3.2.1.1	Comparison of numerically and experimentally obtained lift and drag data	79
3.2.1.2	Spatial convergence tests	81
3.2.2	Dynamic blade tests	86
3.2.2.1	Numerically and experimentally obtained oscillatory aero- foil forces	87
3.2.2.2	Spatial convergence tests	91
4	Turbines at low blockage - Solidity study	94
4.1	Reynolds number = $O(10^5)$	97
4.1.1	Solution convergence	97
4.1.2	Results	104
4.1.2.1	Time-averaged flow fields	106
4.1.2.2	Blade torque	111
4.1.2.3	Instantaneous streamline plots	113
4.1.2.4	Sectional lift and drag forces - indication of dynamic stall	116
4.1.2.5	Turbine torque	123
4.2	Reynolds number = $O(10^6)$	125
4.2.1	Solution convergence	126
4.2.2	Results	128

4.3	Chapter conclusions	138
5	Turbines in confined flow	139
5.1	Flow confinement study	139
5.1.1	Solution convergence	143
5.1.2	Results	151
5.1.2.1	Time-averaged flow fields	152
5.1.2.2	Sectional lift and drag forces	156
5.1.2.3	Blade torque	163
5.1.2.4	Instantaneous streamline plots	164
5.1.2.5	Turbine torque	166
5.1.2.6	Turbine wake	167
5.1.2.7	Overall flow characteristics	171
5.2	Free surface modelling	174
5.2.1	Basin efficiency	180
5.2.2	Froude number dependency	186
5.2.3	Turbine and blade loads	188
5.3	Chapter conclusions	194
6	Conclusions & Future work	195
6.1	Conclusions	195
6.2	Contribution of thesis	200
6.3	Future work	202

List of Figures

1.1	Illustration of positions of sun, moon and earth for spring and neap tides, adapted from Earthsky (2011)	5
1.2	Tidal atlases, adapted from BERR (2008)	7
1.3	Tidal turbine rotor types, taken from Savage (2007)	11
1.4	Examples of unducted axial-flow tidal turbines anchored with a monopile	12
1.5	Further examples of unducted axial-flow tidal turbines using novel mooring systems	13
1.6	Illustration of a streamtube past an axial-flow wind turbine, taken from Burton et al. (2001)	14
1.7	Forces on an actuator disc, adapted from Houlsby et al. (2008)	15
1.8	Energy extraction of a tidal stream turbine, adapted from Houlsby et al. (2008)	17
1.9	Blade element theory (BET) plots, adapted from from (Burton et al., 2001)	18
1.10	Examples of ducted axial-flow tidal turbines	20
1.11	Examples of ducted axial-flow tidal turbines with open centres	23
1.12	Cross-flow turbines	25
1.13	Examples of marine cross-flow turbines	27

1.14	Artist's impression of an array of THAWTs, taken from McAdam et al. (2010)	29
1.15	Examples of cross-flow wind turbine concepts for an urban environment	32
2.1	Illustration of multiple concentric streamtubes, taken from Gaden (2007)	36
2.2	Continuous vs. discrete domain, taken from Bhaskaran and Collins (2011)	52
2.3	Rectangular cell, taken from Bhaskaran and Collins (2011)	54
2.4	Control volume (bold edge) used to illustrate discretisation of a scalar transport equation, adapted from ANSYS Inc. (2009)	55
2.5	Time history of inlet and outlet water depth as well as inlet velocity . .	60
3.1	NACA 0015 blade at $Re_c = 3.6 \times 10^5$: comparison of numerically and experimentally obtained lift and drag coefficients	64
3.2	Computational domain for static blade tests at $Re_c = 3.6 \times 10^5$	69
3.3	Grid convergence tests for static blade simulations at $Re_c = 3.6 \times 10^5$: lift coefficient	71
3.4	Grid convergence tests for static blade simulations at $Re_c = 3.6 \times 10^5$: drag coefficient	71
3.5	Periodic lift and drag histories for a static blade at $\alpha = 30.8^\circ$ using the SST $k - \omega$ turbulence model	73
3.6	Computational domain for rotating turbine tests	75
3.7	Comparison of numerically and experimentally obtained blade torque coefficient traces	77
3.8	NACA 0015 blade at $Re_c = 3.6 \times 10^5$ & 6.8×10^5 : lift and drag coefficients	80
3.9	Computational domain for static blade tests at $Re_c = 6.8 \times 10^5$	81

3.10	Blade resolution regions	82
3.11	NACA 0015 blade at $Re_c = 2 \times 10^6$: oscillating blade tests : lift and drag coefficients	88
3.12	Grid convergence tests for oscillating blade simulations at $Re_c = 2 \times 10^6$	92
4.1	Computational domain for the present turbine solidity investigation . .	96
4.2	Convergence of blade torque and turbine power histories for $B = 2$, $\sigma = 0.019$ and $\lambda = 3$	98
4.3	Wake velocity profiles for $B = 2$, $\sigma = 0.019$ and $\lambda = 3$	99
4.4	Wake velocity profiles for $B = 3$, $\sigma = 0.029$ and $\lambda = 3$	100
4.5	Convergence of blade torque and turbine power histories for $B = 3$, $\sigma = 0.029$ and $\lambda = 3$	101
4.6	Convergence of blade torque and turbine power histories for $B = 4$, $\sigma = 0.038$ and $\lambda = 3$	101
4.7	Wake velocity profiles for $B = 4$, $\sigma = 0.038$ and $\lambda = 3$	102
4.8	Number of revolutions required for convergence in C_P	103
4.9	Power and thrust coefficient variation for varying turbine solidity, σ , at $Re_c = 4.42 \times 10^5$	105
4.10	Time-averaged flow fields at $\lambda = 8$ for three different solidities; $\sigma = 0.019$, 0.029 & 0.038 , together with instantaneous streamlines	107
4.11	Time-averaged flow fields at $\lambda = 3$ for three different solidities; $\sigma = 0.019$, 0.029 & 0.038 , together with instantaneous streamlines	109
4.12	Velocity magnitude comparison at $\lambda = 3$ for three turbine solidities; $\sigma = 0.019$, 0.029 & 0.038	110

4.13 Comparisons of blade torque coefficient, C_m , histories of the 2- and 4-bladed turbines	111
4.14 Instantaneous streamline plots for the 2- & 4-bladed turbines for $86^\circ < \theta < 162^\circ$	114
4.15 Instantaneous streamline plots for the 2- & 4-bladed turbines for $\theta \approx 270^\circ$	115
4.16 Force diagram - cross-flow turbine, adapted from McAdam (2008) . . .	117
4.17 Velocity diagram - cross-flow turbine	117
4.18 Blade coefficient histories for a blade of the 2-bladed turbine at $\lambda = 3$.	119
4.19 Blade coefficient histories for a blade of the 2- & 3-bladed turbine at $\lambda = 8$	121
4.20 Comparison of turbine torque coefficient history at $\lambda = 4$ for $\sigma = 0.019$, 0.029 & 0.038	123
4.21 Convergence of turbine power histories for $B = 2$, $\sigma = 0.019$ and $\lambda = 8$	126
4.22 Wake velocity profiles for $B = 2$, $\sigma = 0.019$ and $\lambda = 8$	127
4.23 Number of revolutions for convergence in C_P	128
4.24 Power and thrust coefficient variation for varying turbine solidity, σ , at $Re_c = 2 \times 10^6$	129
4.25 Comparison of power coefficients for different solidities at $Re_c = 4.42 \times 10^5$ & $Re_c = 2 \times 10^6$	130
4.26 Blade coefficient histories for a blade of the 2-bladed turbine at $\lambda = 3$ operating at $Re_c = 4.42 \times 10^5$ & $Re_c = 2 \times 10^6$	131
4.27 Blade coefficient histories for a blade of the 3-bladed turbine at $\lambda = 3$ operating at $Re_c = 4.42 \times 10^5$ & $Re_c = 2 \times 10^6$	132
4.28 Various comparisons for $\sigma = 0.029$ and $\lambda = 3$ at $Re_c = 4.42 \times 10^5$ & $Re_c = 2 \times 10^6$	133

4.29	Blade coefficient histories for a blade of the 2-bladed turbine at $\lambda = 8$ operating at $Re_c = 4.42 \times 10^5$ & $Re_c = 2 \times 10^6$	134
4.30	Various comparisons for $\sigma = 0.019$ and $\lambda = 8$ at $Re_c = 4.42 \times 10^5$ & $Re_c = 2 \times 10^6$	135
4.31	Blade coefficient histories for a blade of the 3-bladed turbine at $\lambda = 8$ operating at $Re_c = 4.42 \times 10^5$ & $Re_c = 2 \times 10^6$	137
5.1	Computational domain for the present flow confinement investigation .	141
5.2	Convergence of blade torque and turbine power histories for $b = 50\%$ and $\lambda = 3$	144
5.3	Wake velocity profiles for $b = 50\%$ and $\lambda = 3$	145
5.4	Convergence of blade torque and turbine power histories for $b = 12.5\%$ and $\lambda = 2$	146
5.5	Wake velocity profiles for $b = 12.5\%$ and $\lambda = 2$	147
5.6	Convergence of blade torque and turbine power histories for $b = 6.25\%$ and $\lambda = 2$	148
5.7	Wake velocity profiles for $b = 6.25\%$ and $\lambda = 2$	149
5.8	Number of revolutions required for convergence in C_P	150
5.9	Power and thrust coefficient variation for varying blockage, b	151
5.10	Time-averaged flow fields for different λ at blockage, $b = 50\%$, together with instantaneous streamlines	153
5.11	Time-averaged flow fields at λ_{opt} for different blockages together with instantaneous streamlines	154
5.12	Blade coefficient histories at $\lambda = 3$	157
5.13	Blade coefficient histories at $\lambda = 4$	158

5.14	Streamline plots at $\theta \approx 105.0^\circ$ for $\lambda = 3$ and $b = 50\%$ & 6.25% . Note that to ease comparison between cases, each flow field has been rotated such that the blade is traversing vertically downward.	159
5.15	Illustration of effect of curved path of cross-flow turbine blades	161
5.16	Comparisons of blade torque coefficient, C_m , histories	163
5.17	Instantaneous streamline plots for blockages, $b = 50\%$ & $b = 12.5\%$ at $\lambda = 2$ for $120^\circ < \theta < 163^\circ$. Note that to ease comparison between cases, each flow field has been rotated such that the blade is traversing vertically downward.	165
5.18	Comparisons of turbine C_m traces	166
5.19	Turbine wake profiles at maximum power point, λ_{opt}	167
5.20	Comparisons of centreline wake pressures at maximum power point, λ_{opt}	169
5.21	Pressure contour plots for various azimuthal positions, θ , where $b = 50\%$ and $\lambda = 3$	172
5.22	Vorticity magnitude contour plots for various azimuthal positions, θ , where $b = 50\%$ and $\lambda = 3$	173
5.23	Power and thrust curves for different blockages under rigid lid (RL) and deformable free surface (FS) conditions	175
5.24	Wake velocity profiles under rigid lid and deformable free surface conditions for $b = 50\%$ & $\lambda = 3$	177
5.25	Illustration of change in flow depth for different blockages at $Fr = 0.082$	178
5.26	Illustration of free surface deformation due to turbine energy extraction for $b = 50\%$ & $Fr = 0.082$	179
5.27	Schematic of flow mixing states for the rigid lid simulations, adapted from Housby et al. (2008)	181

5.28 Schematic of flow mixing states for the free surface simulations, adapted from Houlsby et al. (2008)	183
5.29 Comparison of basin efficiencies	185
5.30 Froude number dependency	187
5.31 Blade and turbine torque histories for rigid lid and free surface cases . .	189
5.32 Blade coefficient histories for rigid lid and free simulations at $b = 50\%$ and $\lambda = 3$; for the free surface case the blade coefficients are also shown corrected for the effect of the hydrostatic pressure variation across the blade.	191
5.33 Blade torque history for $b = 50\%$ and $\lambda = 3$; for the clockwise case the C_m trace is also shown phase shifted by 180° (corrected).	192

List of Tables

3.1	Percentage error in C_D pre-stall conditions	66
3.2	Changes in lift and drag computations using meshes with different chord-wise discretisations and $y = 1 \times 10^{-5} c$	83
3.3	Changes in lift and drag computations using meshes with different wall normal discretisations	85
4.1	Parameters of a turbine tested at the Sandia National Laboratories . .	95
5.1	Parameters of a turbine tested by McAdam et al. (2010)	139
5.2	Details of the test cases for the flow confinement investigation	142

Acknowledgements

Firstly, I'd like to thank the two supervisors of this project Dr. Richard Willden and Dr. Malcolm McCulloch for the support of my dissertation. In particular, I'd like to extend my gratitude to Dr. Richard Willden for his invaluable advice throughout the study.

Secondly, I'd like to thank my colleagues for their support and helpful opinions; Ross McAdam, Scott Draper, Esteban Ferrer, Ben Twiney, Clarissa Belloni, Simon McIntosh and Conor Fleming.

I'm grateful for the interesting and encouraging discussions I had with my flatmates, Emilio Matthaei and Andreas Glawar, when progress was slow.

Furthermore, I'd like to thank my parents for their endless support over the last few years. Special thanks go to my girlfriend Jane, whose moral support is very important to me. Also, I'd like to extend my thanks to my friend Udo for his generosity.

My appreciations go to the UK Engineering and Physical Sciences Research Council (EPSRC) and the Oxford Martin School for their financial support.

Chapter 1

Introduction

1.1 Outline of thesis

The urgent need to establish a clean, safe and affordable energy supply has placed increased emphasis on the exploitation of new renewable energy sources. The ocean offers immense potential for clean energy extraction and, besides wave power and tidal barrage technologies, tidal stream turbines have been identified as prospective marine energy converters. However, this resource has not yet been exploited and as the underlying flow physics of tidal turbines is not fully understood, fundamental research is still required to ensure tidal energy is harnessed as cost-effectively as is possible. To this end this thesis presents numerical investigations of the hydrodynamics of generic marine cross-flow turbines.

Specifically, the aims of the thesis are to investigate and explore the physics of aspects of cross-flow turbines pertinent to marine application;

- Turbine solidity (ratio of net blade chord to circumference)
- (High) flow blockage (ratio of device frontal area to channel cross-section area) and free surface deformation.

In this chapter, an introduction to renewable and in particular tidal energy is offered. Also, an overview of the technological status as to tidal turbines is given and it is discussed why the present study focuses on horizontal axis cross-flow turbines. Moreover, an overview of the current state of research on cross-flow turbines is given.

In Chapter 2, a presentation of various numerical modelling techniques applicable to tidal turbines is offered and the use of a Computational Fluid Dynamics (CFD) model solving the Reynolds-averaged Navier-Stokes equations for the present study is explicated. Additionally, a detailed description of the pertinent aspects of ANSYS FLUENT 12.0, the software used for the current study, is given.

In Chapter 3 the validation of the CFD model used for the present study is elaborated upon.

In Chapter 4 the first set of results of the present investigation is presented. The effect of turbine solidity on blade forces and turbine performance has been examined as well as the effect of changes in Reynolds number.

In Chapter 5 the second set of results is discussed. Firstly, the effects of flow confinement on turbine performance have been studied as well as the effect of free surface deformation on energy extraction. A volume of fluid approach with novel boundary conditions is used to model the free surface.

Chapter 6 comprises concluding remarks as well as suggestions for future work.

1.2 Renewable & tidal energy

It may be argued that two issues will overshadow future energy supply decisions. Firstly, evidence suggests that global energy generation, currently dominated by fossil fuels, is one of the prime sources for causing or at least accelerating climate change. A strong correlation between the earth's average temperature and the global concentration of carbon dioxide has been demonstrated, see Merchant (2008). Although uncertainty remains over the long-term sustainable level of atmospheric carbon dioxide, there is little doubt that there is a need to significantly reduce our dependency on polluting energy sources, if today's temperature levels and the current eco-system are to be maintained for future generations.

The second key factor is energy security, i.e. how to reliably match demand and supply in years to come. A large gap between the two has been predicted and potential energy shortages have the potential to destabilise the world's political integrity, see Westwood (2008).

These two aspects underline why the renewable energy sector has received renewed attention over the last decade; renewables are expected to play a pivotal role with regard to solving the challenges of climate change and establishing a safe and affordable energy supply. The UK has manifested its support of the renewable energy sector by committing to ambitious goals, such as 20% of the country's electricity to come from renewable sources by 2020, and reducing carbon dioxide emissions based on 1990 levels by 60% by 2050, as outlined in DTI (2003).

While the electricity generated from renewable sources in the UK has increased from 10 TWh/yr in 2000 to 25 TWh/yr in 2010, it is still only about 7% of the UK's electricity generation that came from renewable sources in 2010, see DECC (2011). In order to achieve its goals the UK is required to significantly increase the contribution from renewable sources to the energy supply over the next 10 years.

So far, the focus of investors, developers as well as the media has primarily lain in bio-fuels, wind and solar energy. More recently, however, the vast untapped energy source of the ocean has also caught the interest of the public. Over the last few years, the UK government has made multiple funds available to tidal developers. A large number of independent tidal energy related projects are being carried out in the UK; for further details see Section 1.3.

1.2.1 Tidal dynamics

A detailed analysis of the dynamics of tides may be found in Pugh (1996). Tides are the periodical rise and fall of the surface of the oceans and seas generated by the gravitational attraction and subsequent relative motion of the earth, moon and sun, as described in Nicholls-Lee and Turnock (2008). As stressed in Bryden et al. (2007), the local geography, e.g. the presence of islands or depth variation, may influence the response of the oceanic waters to the astronomic tide generating forces. Both the timing and the magnitude of the tides can be accurately calculated, which, in turn, makes energy generation from tides far more predictable than from other renewable sources that are subject to changeable environmental conditions, e.g. wind or solar. A number of different computational models have been developed to predict the tidal elevations in different parts of the world and how much energy may be removed from tidal basins and channels, e.g. Kawase and Thyng (2010), Draper et al. (2010) and Garrett and Cummins (2005).

When discussing tidal characteristics, one typically differentiates between spring and neap tides. As illustrated in Figure 1.1, spring tides occur when the moon, sun and earth are aligned, whilst neap tides occur when the moon, sun and earth are at right angles. Spring tides, which result in the maximum tidal range, can have twice the tidal range of neap tides, the minimum tidal range, whilst both have a 14 day periodicity, as discussed in Nicholls-Lee and Turnock (2008).

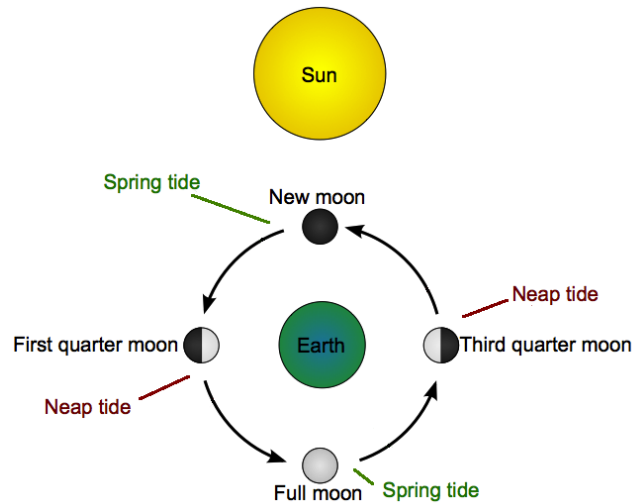


Figure 1.1: Illustration of positions of sun, moon and earth for spring and neap tides, adapted from Earthsky (2011)

1.2.2 Tidal resources

With regard to tidal technologies, one typically distinguishes between tidal range and tidal stream devices.

The tidal range, i.e. the difference in height between high and low tides, can be used to generate electricity by holding water back at high tide and releasing it through turbines at low tide. Tidal barrages are a proven technology, for instance a 240 MW facility has operated in La Rance, France, since 1966. The UK's estimated tidal range potential is 19 TWh/yr, which corresponds to about 4% of the world's tidal range resource, see Savage (2007).

Tidal stream machines are kinetic energy converters and generate electricity by taking advantage of marine currents created by tides. Typically, the flow of water causes parts of the tidal stream device to move and this movement is used to generate electricity. The key advantages of tidal stream turbines over tidal range power generation is their much wider range of available sites as well as their reduced environmental impact, e.g. elaborate and costly barrages are avoided. The total UK tidal stream resource

is estimated to be 110 TWh/yr by Black & Veatch (2005). However, the technically extractable energy potential due to tidal streams around the British Isles is predicted to be 18 TWh/yr, which equates to about 12% of the global extractable resource, see Savage (2007). Black & Veatch (2005) consider 12 TWh/yr to be economically exploitable with current technologies, which equates to about 3% of the UK's electricity consumption in 2010, see DECC (2011).

1.2.3 Potential sites for tidal stream energy generation in the UK

One of the key criteria when selecting a suitable location for the deployment of tidal stream turbines is naturally the power available for energy extraction. Figure 1.2a shows a map of the UK illustrating the average tidal power and it is apparent that there are a number of highly energetic tidal sites around the British Isles.

One location that has received particular interest with regard to exploiting tidal stream energy is the Pentland Firth on the northern coast of Scotland; Dillon and Woolf (2008) calculated the total energy available in the Pentland Firth to be 3 GW, which equates to 3.6% of Britain's current total power capacity.

Figure 1.2b shows an atlas of the UK illustrating the peak flow velocity of a mean spring tide and the largest peak flow velocities are observed in the Pentland Firth; the mean spring peak velocity is the mean of the highest velocity of tidal movement that can be found in an area during a lunar month.

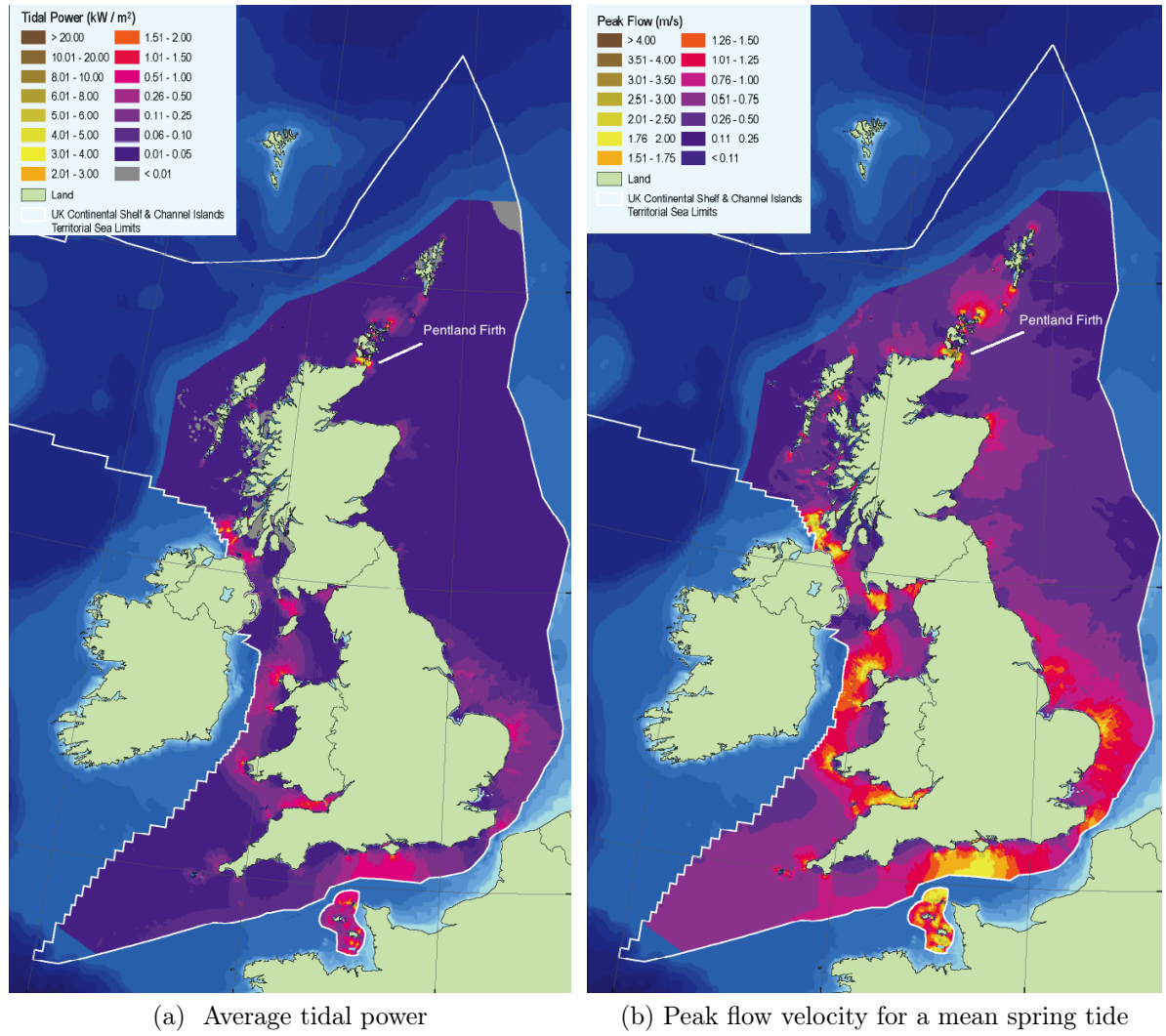


Figure 1.2: Tidal atlases, adapted from BERR (2008)

In Nicholls-Lee and Turnock (2008), it is discussed that a mean spring peak current of more than 2 m/s is required for an economically viable electricity generation from tidal stream turbines. Black & Veatch (2005) suggest that 53% of the technically extractable resource is at sites with mean spring peak velocities larger than 3.5 m/s and 28% of the technically extractable resource at sites with mean spring peak velocities higher than 5.5 m/s .

However, in order to examine the suitability of a tidal basin or channel for energy generation using tidal stream turbines, a number of factors in addition to the peak

velocities and power available are to be considered.

The variation in flow velocity over one (lunar) day, which comprises a cycle of two highs and two lows, as well as the difference in flow velocity between spring and neap tides are important criteria. For instance, Dillon and Woolf (2008) found that flow velocities in the area of peak flow in the Pentland Firth are larger than 1 m/s for 80% and larger than 2 m/s for 55% of the time. This is advantageous with regard to being able to generate electricity for the majority of time, but such conditions will allow for minimal time to perform turbine installations and maintenance works.

Further aspects are possible conflicts with other sea users as well as the impact of tidal stream turbines on marine flora and fauna. Moreover, the installation domain should avoid significant exposure to open seas, waves and winds in order to reduce the risk of weather induced delays and for ease of installation and servicing, as summarised in Nicholls-Lee and Turnock (2008).

The present author believes that the Pentland Firth will not be among the locations, where tidal stream turbine farms will be deployed in the near future. Due to its high potential power output, see Dillon and Woolf (2008), the Pentland Firth is of great interest for tidal turbine developers, but its harsh and unaccommodating environment do not render it suitable for tidal turbine farms early in the industry's development cycle. There are other locations, which offer longer periods of low flow velocities and lower water depths than the Pentland Firth, which will render the installation procedure and maintenance works significantly less challenging.

One example is the Strangford Lough in Northern Ireland, where Marine Current Turbines (MCT), for more details on the turbine see Section 1.3.1, installed their first full scale device, Seagen. The maximum current velocity at the site was measured to be 4.8 m/s , the depth of the water mean sea level is 26.2 m , the rated speed of the turbine is 2.4 m/s and the turbine has a rated power of 1.2 MW, which results in an expected energy generation of 3800 MWh/yr, see Marine Current Turbines (2011).

Black & Veatch (2005) also anticipate that sites with water depth, $h = 30 - 40$ m, are likely to be targeted first by tidal developers. In the long term, however, it appears inevitable for the technology developers to focus on devices suitable for depths of more than 40 m. According to the results presented in Black & Veatch (2005), only 30% of the technically extractable resource (18 TWh/yr) is at sites with $h = 30 - 40$ m, whilst 63% are estimated to be at sites with depths greater than 40 m. This is of particular importance when investigating how economic tidal stream energy generation may be, as advantages arising from economies of scale are anticipated to lead to significant reductions in tidal stream energy costs.

1.2.4 Cost of tidal stream energy

The total cost of tidal stream energy depends on a large number of factors, for example:

1. flow velocity distribution over one day;
2. direction of flow; some tidal sites are almost bi-directional, but, as discussed in Myers and Bahaj (2005) and Blunden and Bahaj (2006), many highly energetic tidal sites exhibit flow reversal of 20° or more away from 180° ; for instance, at Portland Bill, UK, a flow reversal of 35° from rectilinearity has been observed.
3. water depth, which will affect the type of mooring of the turbine;
4. distance from shore & electrical connection; tidal stream energy generation is in some ways similar to and in direct competition with off-shore wind generation. The tidal technology developer Atlantis Resources Corporation has suggested that the size of a tidal turbine farm may be up to 10 times smaller than the size of an off-shore wind turbine farm for the same power output, see Atlantis Resources Corporation (2011). With regard to the cabling, which is expected to contribute significantly to both wind and tidal turbine farms' capital cost, the reduction in a farm's size by a factor of up to 10 may offer substantial advantages.

5. availability of construction vessels;
6. geotechnical properties.

As commercial tidal turbine farms are yet to be developed and deployed, it is difficult to predict an accurate cost of tidal energy extraction. Because of large uncertainties, for instance the optimum rotor design or the most cost-effective mooring approach are still in question, cost projections are essentially speculative.

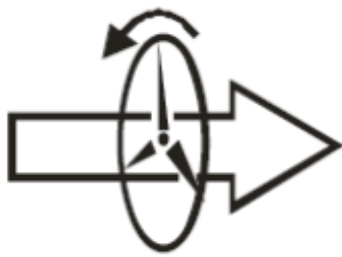
However, the Carbon Trust has carried out an extensive assessment to estimate capital costs as well as the operation and maintenance costs of tidal turbine farms. In Black & Veatch (2005), it is anticipated that the first large-scale tidal stream turbine farms have an energy cost of about 7 p/kWh. Moreover, it is suggested that as 1000 MW of installed capacity are reached, the cost of energy may reduce to 5 p/kWh and that at the highest velocity sites an energy cost of 3 p/kWh may be reached, which is expected to be competitive with the wholesale electricity price. The study by DTI (2001) came to similar conclusions regarding the potential cost of tidal stream energy; it was estimated that farms with an installed capacity of 30 MW operating at a water depth of 30 *m* and a mean spring peak velocity of 3 *m/s* could provide electricity at 4-6 p/kWh.

In summary, tidal stream energy is a renewable and highly predictable energy source that is estimated to potentially contribute up to 5% of the UK's electricity supply at a competitive cost. However, the tidal stream turbine industry is still at an early stage in its development cycle, only a small number of full-scale devices have been deployed and a commercial tidal turbine farm is still to be installed.

1.3 Overview of technological status

In contrast to the wind industry, where the axial-flow turbine has emerged as the predominant energy converter, the tidal stream sector remains uncertain about its best approach to harness the oceans' energy. Different designs have different use and so far it is not evident, which device type will prove to be the most cost-effective.

The two most common types of turbine focused upon by developers of tidal stream devices are axial- and cross-flow machines. As shown in Figure 1.3, the rotors of the former typically prescribe a circle and the undisturbed current, from which energy is extracted, is aligned with the axis of turbine rotation. In contrast, the axes of cross-flow turbines are perpendicular to the flow and the blades move across the flow as the rotor turns.



(a) Axial-flow turbine



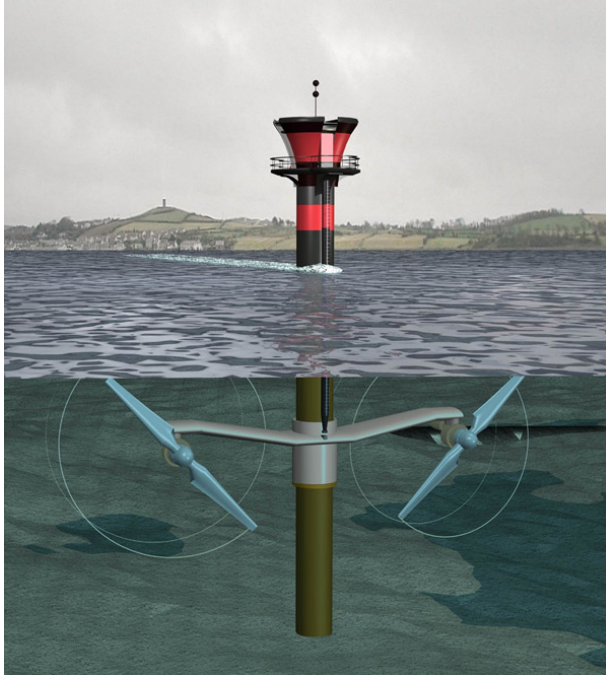
(b) Cross-flow turbine

Figure 1.3: Tidal turbine rotor types, taken from Savage (2007)

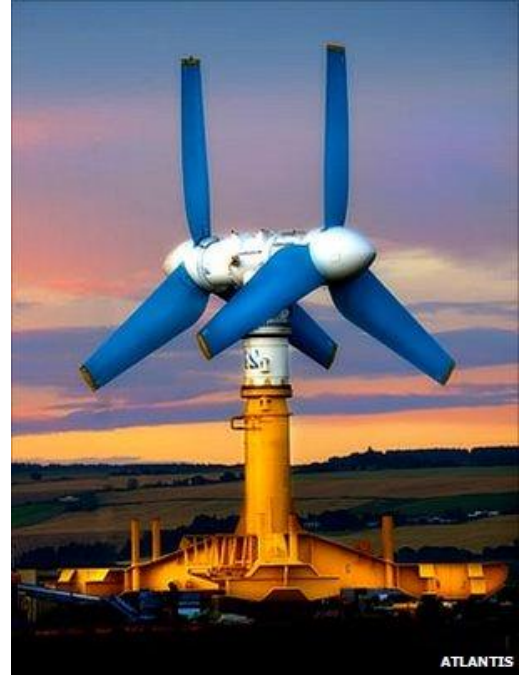
1.3.1 Axial-flow turbines

1.3.1.1 Unducted turbine designs

Axial-flow turbines are a proven technology from the wind industry. Examples of tidal axial-flow turbines include those developed by Marine Current Turbines (MCT), see Figure 1.4a, Atlantis Resources Corporation (ARC), see Figure 1.4b, Hammerfest Strøm, see Figure 1.5b, Lunar Energy, see Figure 1.10a, Clean Current, see Figure 1.11a, Open-Hydro, see Figure 1.11b, Swanturbines (2011) and TidalStream (2011).



(a) MCT turbine - SeaGen, taken from Marine Current Turbines (2011)



(b) ARC turbine, taken from Atlantis Resources Corporation (2011)

Figure 1.4: Examples of unducted axial-flow tidal turbines anchored with a monopile

Both MCT and ARC rely on a monopile on which two axial-flow turbines operate. As shown in Figure 1.4a, MCT positions the two rotors in an adjacent arrangement, which results in increasing the area covered by the blades and hence in increasing the power generated per monopile. In order to accommodate for the reversal in tidal flow direction, the MCT design incorporates a variable blade pitch mechanism. In contrast, the rotors of the ARC turbine are arranged in anterior-posterior positions, where the downstream rotor is always kept stationary. This configuration allows ARC to use fixed pitch blades; as the tidal flow direction reverses, the previously stationary turbine starts to rotate and generate energy.

Another variant of the axial-flow turbine is being developed at the University of Strathclyde. As for the ARC machine, the turbine developed at the University of Strathclyde comprises two rotors in an anterior-posterior arrangement, see Figure 1.5a. However, these rotors are contra-rotating, which the developers expect to lead to a minimisation

of reactive torque transmitted to the support structure, as discussed in Clarke et al. (2008). This may permit the use of a relatively simple and economic mooring system, for instance by flying the turbine from a tension cable.

In fact, the choice of optimum rotor for a tidal stream turbine is largely dependent upon a respectively cost-effective mooring system. Hammerfest Strøm are also developing an axial-flow turbine, but are considering a tripod-like arrangement with regard to the mooring, see Figure 1.5b. It remains to be seen, which design will be the most cost-effective approach to generate electricity from tidal streams.



(a) Contra-rotating marine turbine, taken from Clarke et al. (2008)



(b) Hammerfest Strøm turbine, taken from Hammerfest Strom (2011)

Figure 1.5: Further examples of unducted axial-flow tidal turbines using novel mooring systems

As mentioned above, one of the significant advantages of axial-flow turbines is that they are a proven technology from the wind turbine industry. In fact, much research has been carried out relating to the aerodynamics of wind turbines and although crucial differences exist to marine applications, the theories developed for the wind industry render a useful starting point to understand and analyse the hydrodynamics of tidal stream (axial-flow) devices. Below, a short introduction into a simple modelling ap-

proach for energy extraction using axial-flow turbines as well as a brief discussion of the main difference between wind and tidal stream generation is presented in order to allow for a more profound comparison of the different rotor types. A detailed account of various modelling techniques of tidal turbine flows is offered in Chapter 2.

1.3.1.2 Momentum actuator disc concept

The one-dimensional single actuator disc theory presented in Newman (1983) is a method to identify the maximum power an axial-flow wind turbine can generate. The following assumptions are made:

1. incompressible, inviscid and steady flow;
2. unbounded fluid domain;
3. infinite number of blades – circular disc over which the axial force is uniformly distributed;
4. uniform velocity over the disc.

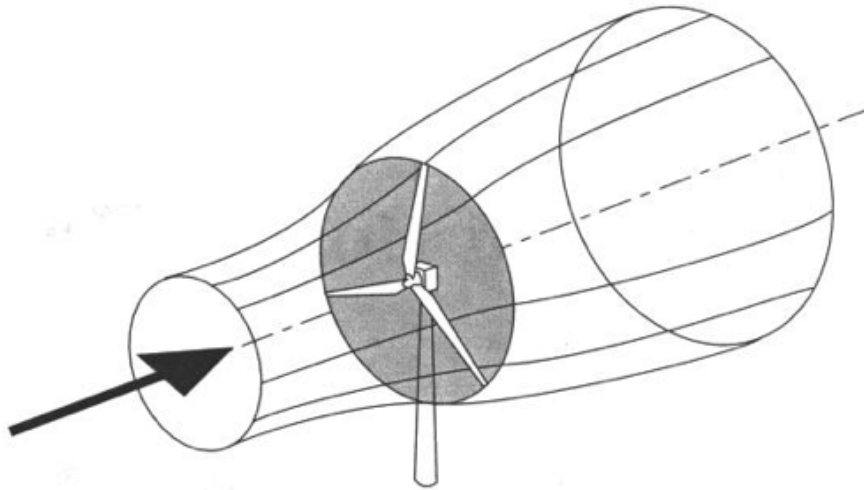


Figure 1.6: Illustration of a streamtube past an axial-flow wind turbine, taken from Burton et al. (2001)

Moreover, it is assumed that the air that passes through the rotor remains separate from the air that does not, which allows us to draw the bounding streamtube as shown in Figure 1.6. By continuity, a reduction in flow speed, due to the extraction of kinetic energy, corresponds to an increase in the tube's cross-sectional area, which explains the shape evident in Figure 1.6. The actuator disc theory treats the turbine as a permeable disc which experiences a pressure difference as the flow passes through it, see Figure 1.7. By applying continuity, conservation of axial momentum and Bernoulli's equation both upstream and downstream of the disc, it can be shown, see Burton et al. (2001), that the force T acting on the air is equal to:

$$T = 2\rho AU_\infty^2 \alpha_2(1 - \alpha_2) \quad (1.3.1)$$

where U_∞ denotes the free stream velocity, α_2 the turbine velocity coefficient, ρ the fluid's density and A the cross-sectional area of the disc.

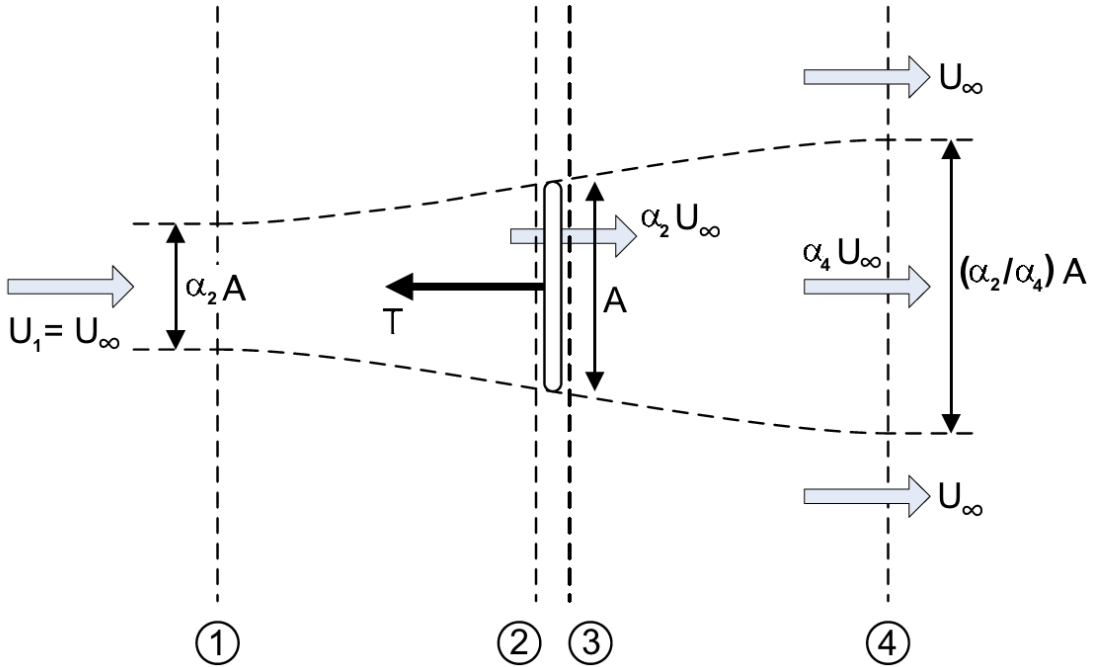


Figure 1.7: Forces on an actuator disc, adapted from Houlby et al. (2008)

The power coefficient C_P , a measure of efficiency, is defined as follows, in which it should be noted that the total kinetic flux through the projected area of the turbine is equal to $\frac{1}{2}\rho AU_\infty^3$:

$$Power = Force \times Velocity = T\alpha_2 U_\infty = 2\rho AU_\infty^3 \alpha_2^2 (1 - \alpha_2) = \frac{1}{2}\rho U_\infty^3 AC_P \quad (1.3.2)$$

By simple differentiation with respect to α_2 , it can be shown that $\max C_P$ is equal to $16/27$ - the well-known Betz limit, see Betz (1920), which states that it is theoretically impossible to extract more than 59.3% of the total (kinetic) energy available.

However, a marine cross-flow turbine tested by McAdam et al. (2010) attained efficiencies of up to 160%, using the same definition as presented in Equation 1.3.2. This deviation from the Betz limit stems from the fact that the flow of air in the atmosphere is fundamentally different from the marine tidal case, as discussed in Bryden et al. (2007) and Whelan et al. (2009). The Betz approach assuming an unbounded flow, where the boundaries are infinitely distant from the energy converter, is unrealistic for a tidal application with significant blockage. The flow confinement due to the proximity of the seabed and sea surface causes higher flow velocities through the actuator plane than postulated by Betz's theory. Hence, the energy extraction potential of a bounded flow turbine can be greater than Betz's theoretical limit.

Moreover, the fact that the free surface, i.e. the interface between water and air, is deformable indicates that it is static head (the sum of pressure head and potential head) and not kinetic head that is extracted. In fact, a marine turbine converts static head into kinetic head, of which some proportion, depending on the efficiency of the turbine, is extracted and used to generate electricity. The kinetic head across the turbine increases. By open channel flow theory, the extraction of energy from a sub-critical flow leads to a reduction in the flow depth and hence, by continuity, to an increase in fluid

velocity, see Figure 1.8.

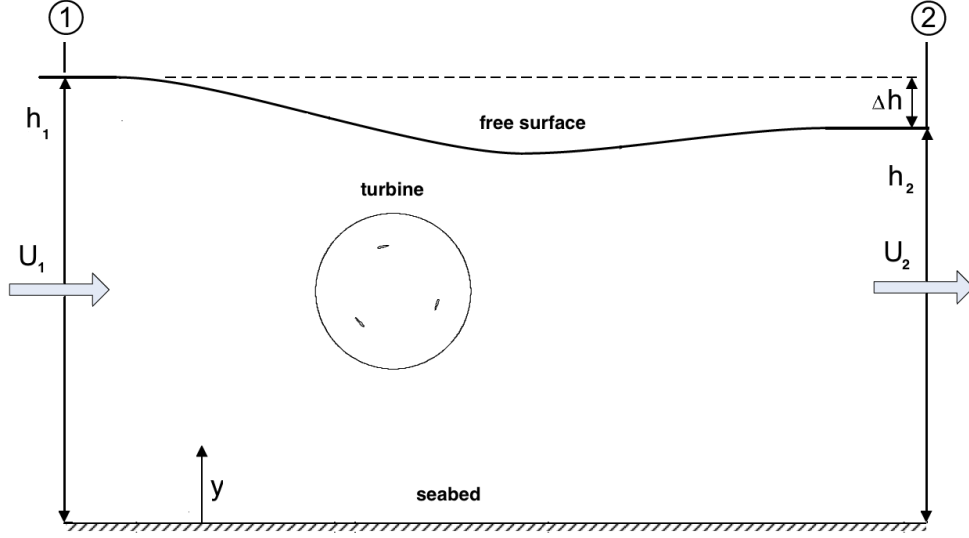


Figure 1.8: Energy extraction of a tidal stream turbine, adapted from Houlsby et al. (2008)

The maximum power (P_{max}) a tidal turbine can extract is equal to the difference in total head upstream and downstream of the machine:

$$P_{max} = \dot{m}g \left(h_1 + \frac{U_1^2}{2g} \right) - \dot{m}g \left(h_2 + \frac{U_2^2}{2g} \right) = \dot{m}g (H_1 - H_2) \quad (1.3.3)$$

where $\dot{m}$ is the mass flow rate, g gravity, h the flow depth, U the velocity and H the total head.

Also, Houlsby et al. (2008) have extended the one-dimensional momentum actuator disc theory to a general cross-sectional array of tidal turbines in an open channel tidal flow. They found that the efficiency of an arbitrary array of turbines can be determined relative to the total power extracted from the channel flow. Moreover, it is suggested that a general form of this efficiency may be more important to characterise the performance of tidal stream turbines than the typical power coefficient, as defined in

Equation 1.3.2, used for wind turbines, as the total power that may be extracted from a channel flow is likely to be limited by environmental constraints. This is discussed further in Section 5.2.

It is to be noted that Equation 1.3.3, just as Equation 1.3.2, only provides information about the maximum theoretical power, but little on the actual power generated, which is determined by the rotor performance.

1.3.1.3 Blade element theory

Blade element theory (BET), which accommodates for the loading of individual blades, allows the designer to estimate how much power a turbine will actually generate. BET was originally developed to determine the behaviour of propellers, see Weick (1926); today, it is commonly used in the wind turbine and helicopter industries to derive the (aerodynamic) forces generated by rotor blades.

Each blade is discretised in the spanwise direction into a set of two-dimensional (2D) blade elements, on which aerodynamic analysis is performed individually and by spanwise integration, the overall thrust and torque can be defined. Figure 1.9a & 1.9b illustrate an example of an axial-flow wind turbine, for which a simplified BET calculation is shown.

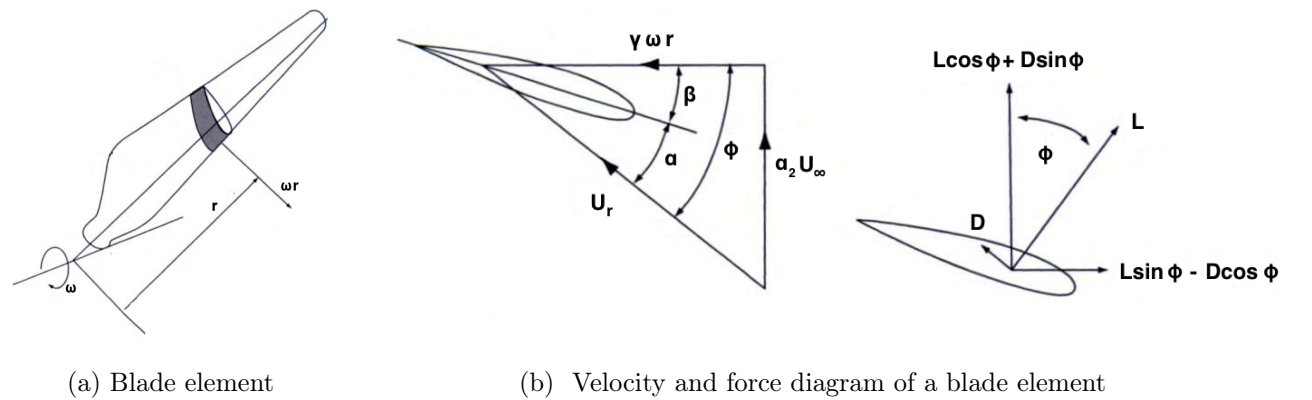


Figure 1.9: Blade element theory (BET) plots, adapted from from (Burton et al., 2001)

As the fluid flow passes the blade, it is diverted from its original path leading to changes in the speed of the fluid and consequently, by Bernoulli's principle, the pressure exerted by the fluid on the blade is altered from that of the undisturbed flow. In addition, the viscosity of the flow generates frictional forces, resisting the flow around the blade. Hence, as a result of the change in pressure and the frictional forces, the blade experiences a resultant hydrodynamic force, which is typically separated into lift, a force perpendicular to, and drag, a force parallel to the incident flow vector. The global thrust, T , torque, Q_T , and power, P , are calculated as follows, see Figure 1.9b:

$$T = B \int_{root}^{tip} (L \cos \phi + D \sin \phi) dr \quad (1.3.4)$$

$$Q_T = B \int_{root}^{tip} r (L \sin \phi - D \cos \phi) dr \quad (1.3.5)$$

$$P = Q_T \omega \quad (1.3.6)$$

where L and D are the sectional lift and drag forces, B is the number of blades, r the radius, ω the angular velocity, ϕ denotes the flow angle, i.e. the angle between the resultant flow velocity relative to the blade and the plane of blade rotation; also, β in Figure 1.9b is the blade's geometric incidence relative to the rotation plane, such that α is flow's angle of attack relative to the blade chord line, and:

$$L = \frac{1}{2} \rho U_r^2 c C_L \quad (1.3.7)$$

$$D = \frac{1}{2} \rho U_r^2 c C_D \quad (1.3.8)$$

where c is the chord length of the blade and U_r the resultant velocity:

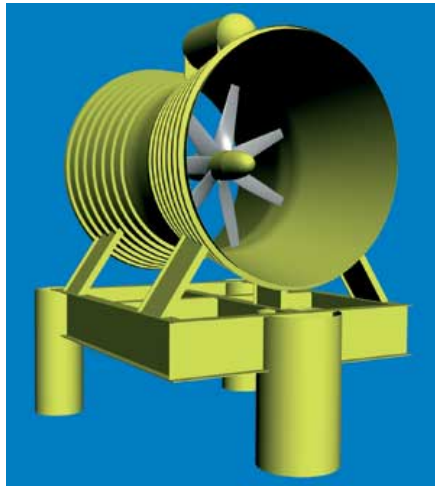
$$U_r^2 = (\alpha_2 U_\infty)^2 + (\gamma \omega r)^2 \quad (1.3.9)$$

where U_∞ is the free stream velocity, α_2 the axial velocity induction coefficient at the rotor plane and γ is the circumferential velocity induction coefficient at the rotor plane; the latter is due to the reaction torque imposed upon the air, which causes the air downstream of the turbine to rotate in the opposite direction to the rotor, see Burton et al. (2001).

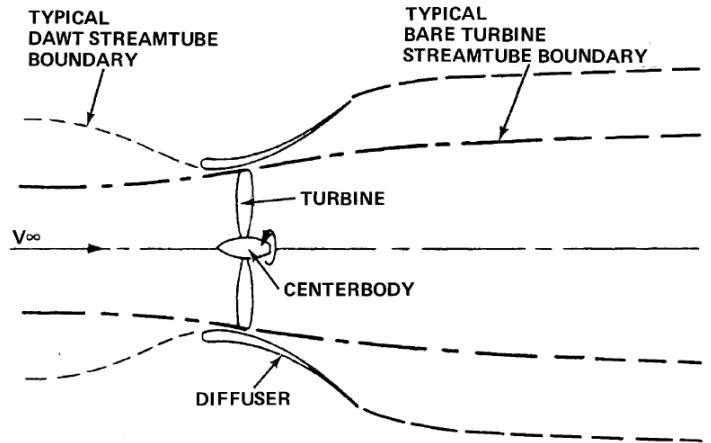
The values of the lift and drag coefficients C_L and C_D are obtained from experimental data, corresponding to the particular flow condition, i.e. angle of attack, Reynolds Number etc., of each specific blade element.

1.3.1.4 Ducted turbine designs

But not all marine axial-flow turbines resemble the conventional modern windmill, discussed above. For instance, Lunar Energy have added a duct around their rotor, see Figure 1.10a.



(a) Lunar energy turbine, taken from Lunar Energy (2011)



(b) Illustration of the effect of diffuser augmented turbines, taken from Foreman (1981)

Figure 1.10: Examples of ducted axial-flow tidal turbines

Using a duct can have numerous advantages, for example improved safety, protection from weed growth and a reduced turbine size for a given power output, as discussed in Kirke (2005).

As shown in Figure 1.10b, a duct is typically designed so that the downstream side acts as a diffuser, which leads to an increase in the area of the streamtube downstream of the rotor and hence, by continuity, a decrease in the downstream velocity and thus a decrease in total pressure P_0 at the outlet, where:

$$P_0 = P + \frac{1}{2}\rho U^2 \quad (1.3.10)$$

$$\therefore P_{0_{outlet}} = P_\infty + \frac{1}{2}\rho U_{outlet}^2 \quad (1.3.11)$$

where P and U are the static pressure and velocity at the point for which P_0 is to be evaluated; P_∞ is the free stream static pressure and U_{outlet} the velocity at the outlet.

Assuming identical flow velocities at the inlet and across the rotor, a lower total pressure for the diffuser augmented turbine implies, by Bernoulli, that the static pressure on the downstream side of the turbine disc has to be lower, too. Hence, more flow is drawn through the rotor, thus increasing the power take-off for a given sized turbine.

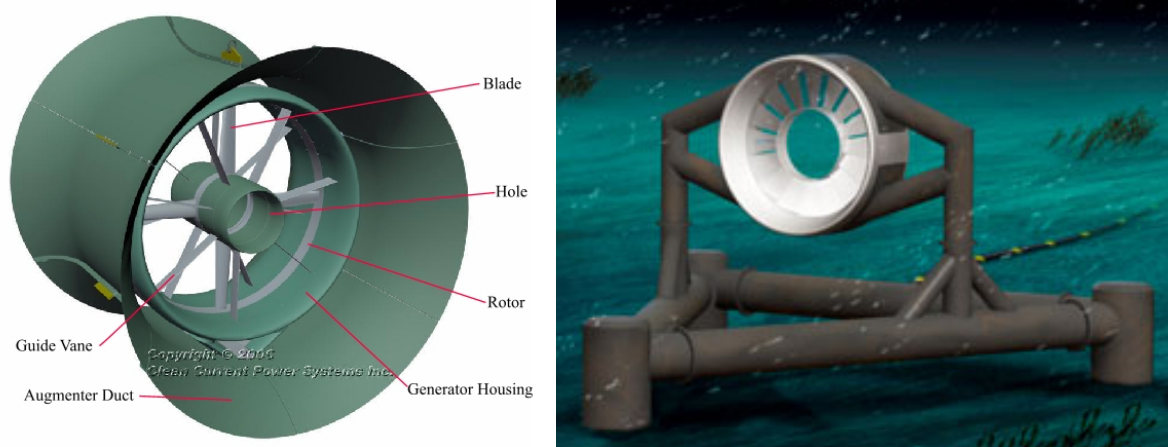
In addition, a smaller turbine in a faster flow spins faster, so that the torque for a given power output is less, see Equation 1.3.6, and consequently a smaller and lower cost gearbox can be used. This is particularly important for tidal turbines, since flow velocities are typically low and consequently large torques are required to generate power.

Despite these benefits, however, ducted turbines are rarely used for wind applications, because they have proven uneconomic for larger and more productive turbines. Since Power $\propto$ Area, see Equation 1.3.2, Power $\propto L^2$, where L is the characteristic length

scale; in contrast, $\text{Costs} \propto \text{Volume}$ and thus $\text{Costs} \propto L^3$. This indicates that there lies a maximum size for which all turbines, diffuser augmented or otherwise, are cost-effective and it has been argued that the additional costs for ducts of conventional sized windmills outweigh the advantages, for example see Doerner (1975) and van Bussel (2007).

However, with regard to tidal applications, there are numerous developers, e.g. Lunar Energy, Clean Current and OpenHydro, who believe that ducts offer additional advantages for tidal turbines, which will offset the supplementary costs etc. For instance, rather than incorporating a mechanism, which allows to fine tune a machine's orientation depending on the direction of flow, ducts can be used to accommodate for changes in the yawing angle. Lunar Energy (2008) explained that their turbine works as efficiently as for the perfectly aligned case, if not better, for incoming flow angles less than 40° .

Furthermore, a duct eliminates tip losses, thus improving the efficiency, and can also be beneficial for protecting the blades. By mounting the hydrofoils on both an inner ring and an outer ring/duct, for examples see Figure 1.11, the stresses in the blades are reduced, provided the outer ring's centrifugal acceleration, which would increase the axial stresses, is minimal. Due to the high (fatigue) loads marine turbines are expected to be able to withstand, a decrease in the blades' stresses is particularly important. It is for this precise reason that the turbines of both Clean Current and OpenHydro have a hole in the middle, as shown in Figure 1.11. Lift, the key driver for torque, see Equation 2.1.2, $\propto U_r^2$ and $U_r^2 \propto r^2$, see Equation 1.3.9, and thus little torque is contributed by the blade elements close to the centre. The reduction in stresses due to the shorter blades might hence prove advantageous, as weaker, i.e. cheaper, materials can be used.



(a) Clean Current turbine, taken from Clean Current (2011) (b) OpenHydro turbine, taken from OpenHydro (2011)

Figure 1.11: Examples of ducted axial-flow tidal turbines with open centres

The computational study of open-centre tidal turbines discussed in Belloni and Willden (2011) showed that the power produced by a turbine, for which the ratio of the diameter of the open centre and the rotor was $3/13$, was about 2.5% smaller than the power generated by the corresponding turbine without the open centre. However, any further increases in the diameter of the open centre resulted in significant reductions in the power output due to the decrease in turbine annulus area from which power can be extracted from the flow.

Moreover, Belloni and Willden (2011) compared the performance of ducted and un-ducted turbines. The results showed a decrease in power output for all ducted turbines, with and without open centre, relative to a bare turbine of the same external diameter. Future field tests will need to show whether the potential benefits of a ducted turbine outlined above may outweigh the negative effects of ducts.

1.3.1.5 Turbine solidity

One of the key differences between the designs of Clean Current and OpenHydro turbines is the large difference in turbine solidity, defined in Equation 3.1.7, i.e. the area

covered by blades, which can have a significant influence on the device's performance.

The power output of a turbine is proportional to the thrust that it exerts onto the flow, see Equation 1.3.2, and the larger the thrust, the larger the flow resistance the turbine presents to the flow, see Equation 1.3.1. The maximum C_P for any wind turbine, the so-called Betz limit, which is discussed above, lies in between two extremes, i.e. a very high and a very low impedance – both result in minimal power extraction. For the case of a very high impedance, little fluid flows through the turbine, instead it is diverted around the machine; for the case of a very low impedance, the turbine hardly affects the flow. In order that the turbine presents an “efficient” impedance to the flow, the right balance between its solidity, σ , and tip speed ratio, λ , needs to be found, where:

$$\sigma = \frac{cB}{\pi 2R} \quad (1.3.12)$$

$$\lambda = \frac{\omega R}{U_\infty} \quad (1.3.13)$$

where c is the chord length of the blade, B denotes the number of blades, R is the radius at the blade's tip, ω the angular velocity and U_∞ the free stream velocity.

If the solidity is high, as for the OpenHydro machine, the operating tip speed ratio has to be low in order to avoid too high an impedance, which has numerous implications.

Firstly, a low tip speed ratio implies increased torque, see Equation 2.1.3, so that a larger and higher cost gearbox has to be used. Secondly, the lower the tip speed ratio, the higher the maximum angle of attack, which can be problematic with regard to blade stalling; from Figure 1.9b it is evident that the larger the ratio of free-stream velocity to rotor circumferential velocity, the larger the angle of attack, α .

However, it is hard to predict the precise effects of the various phenomena, which is why it is difficult to comment on the overall performance of a particular machine. This

eludes to why so many significantly different designs are still being pursued rather than focusing on the optimisation of one established approach, as in the wind industry.

1.3.2 Cross-flow turbines

The most common example of a cross-flow turbine is the Darrieus machine, see Figure 1.12a. Various different planforms are possible, but tidal cross-flow turbines typically present a square cross-section to the flow (cf. circular frontal area of axial flow devices). This has the advantage of covering a larger projected frontal area. Darrieus turbines are multi-directional by nature, meaning that the device's functionality/performance is independent of the direction of flow.

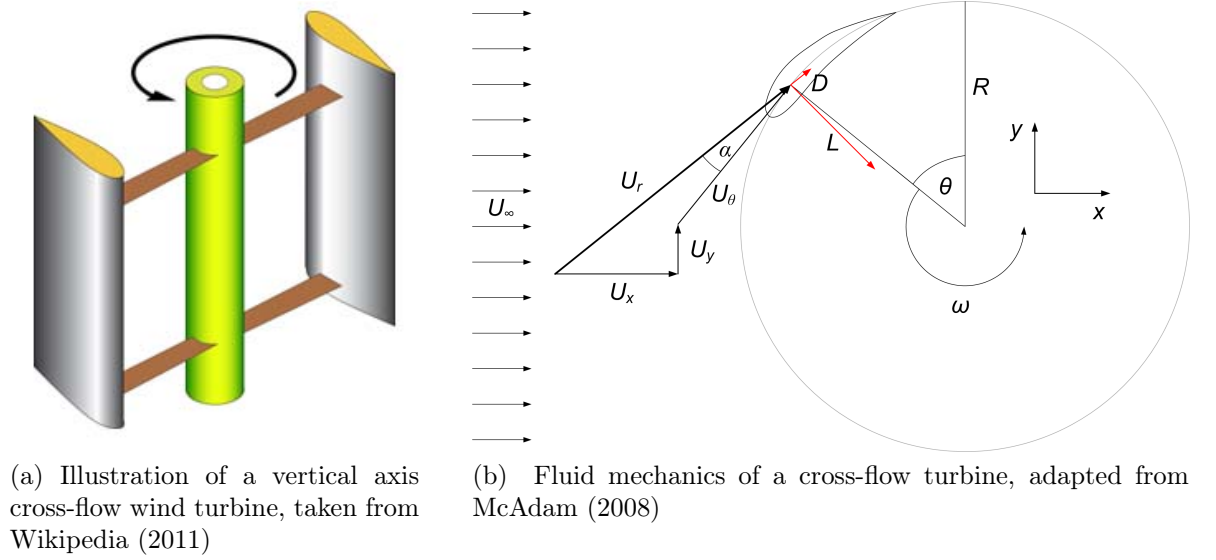


Figure 1.12: Cross-flow turbines

Figure 1.12b outlines the fluid mechanical principles of a cross-flow turbine. The incident flow velocity, U_∞ , once modified by induction factors to yield a local velocity vector $(U_x, U_y)^t$, can be superimposed on the blade rotational velocity, U_θ , to yield the resultant flow velocity relative to the blade, U_r , which forms an angle of incidence α with the blade chord line. The circumferentially resolved components of the resulting

lift and drag forces are the drivers behind a cross-flow turbine, as they combine to give the resultant torque per unit length of blade, Q , about the axis of turbine rotation:

$$Q = BR(L \sin \alpha - D \cos \alpha) \quad (1.3.14)$$

where B , R , L and D are the number of blades, radius at the blade's tip and sectional lift and drag forces.

It should be noted that positive torque continues to be generated throughout the entire cycle of rotation, as long as $L \sin \alpha$ is greater than $D \cos \alpha$. As the blade goes through a rotation of 180° α becomes negative and consequently, for a symmetric blade, the direction of lift reverses relative to the blade, but the direction of the resultant lift is always such as to produce a positive torque. The key challenge due to the change of the lifting surface is to design a hydrofoil, which generates high lift to drag ratios for both positive and negative angles of attack, which is why many cross-flow turbines employ symmetric blades.

Due to the symmetry of the turbine and blades a change in flow direction results in a 180° azimuthal phase shift in the flow vectors and resultant hydrodynamic forces. A cross-flow turbine always rotates in the same particular direction, defined by the orientation of the blades' leading and trailing edges, and is thus independent of the current's direction.

When comparing cross-flow turbines with axial-flow devices, experience from the wind industry suggests that the latter are in principle more efficient. This is due to destructive interference effects of the front row of blades, so that the downstream half of a cross-flow turbine sees a significantly disturbed flow, e.g. more turbulent and slower, thus carrying less kinetic energy.

However, cross-flow turbines offer a number of advantages over more conventional axial-flow turbines, which may outweigh the losses due to the negative downstream wake

effects. The volume of tidal flow intercepted by a rectangular fronted cross-flow turbine is greater than that intercepted by a circular fronted axial-flow turbine of the same diameter. Hence, cross-flow turbines have greater theoretical potential for energy extraction, both as they intercept a greater energy flux in the undisturbed stream, but also as they present a greater effective blockage and can therefore, when placed in a constrained stream, act to force more flow through the turbine, see Whelan et al. (2009). Furthermore, the modular design distinctive to cross-flow turbines permits the formation of single long turbine arrays, which enable devices to act collectively as barrages with increased performance over that of single devices due to their mutual flow blocking effect. Moreover, such turbine arrays may also allow for a reduction in installation and maintenance costs.

Examples of developers working on cross-flow devices include GCK Technology, see Figure 1.12a, Ponte di Archimede International (2011) as well as a research group at the University of Oxford, of which the present author is a member.



(a) Gorlov turbine, taken from GCK Technology (2011)



(b) Transverse Horizontal Axis Water Turbine (THAWT)

Figure 1.13: Examples of marine cross-flow turbines

GCK Technology’s device shown in Figure 1.13a is called the Gorlov turbine, for more information see Gorlov (1998) and Gorlov (2003). Its key innovation is to use twisted blades that shape into a helix, aimed at increasing the structure’s stability. Due to the blade’s twist the hydrofoils are more evenly distributed through the cycle, so that the spanwise integral forces do not change as abruptly as the local lift and drag forces. Consequently, the torque curves produced are smoother and thus the levels of vibration and noise reduced. However, due to its low stiffness this design is axially weak and hence not suited for combining multiple turbines.

In response to the common Darrieus turbines’ lack of axial strength, the concept of a truss structure for a Transverse Horizontal Axis Water Turbine (THAWT), shown in Figure 1.13b, was developed by Houlby, McCulloch and Oldfield, see McAdam (2007) and McAdam et al. (2010).

1.3.2.1 Transverse Horizontal Axis Water Turbine (THAWT)

As the name suggests, a THAWT’s axis is not vertical, as for most cross-flow turbines, but horizontal and the fluid flows transversely to the axis of rotation through the machine, see Figure 1.13b. This arrangement offers numerous advantages:

Similar to the Gorlov turbine, a THAWT can be fixed at both ends, which leads to a reduction in stresses and stabilises the structure. In addition, numerous machines can be combined to form an array, which is crucial to the success of the THAWT concept. Because of material constraints and because the seabed’s depth sets a practical limit on the maximum diameter, it is important to find a solution, which allows for intercepting a greater volume of tidal flow without increasing the diameter. A practical and cost effective solution appears to be to stretch the turbine in a horizontal direction, i.e. forming an array of THAWTs, see Figure 1.14. In contrast to installing a large number of individual (axial-flow) turbines, the costs can be reduced considerably, as fewer piles/foundations, generators etc. are required.

However, a significant increase in stresses is associated with linking a number of turbines together. A solution to this problem may be a truss configuration, see Figure 1.13b, which McAdam (2007) showed to reduce stresses by a factor of 2.7 at minimum and thus allowing for longer arrays.

Today, it remains uncertain whether a tidal cross-flow turbine concept may successfully compete with axial-flow designs. The work conducted for the present study is aimed at furthering the ability to assess the economic feasibility of cross-flow turbines for tidal energy generation. The section below offers a summary of the current state of the research on cross-flow turbines.

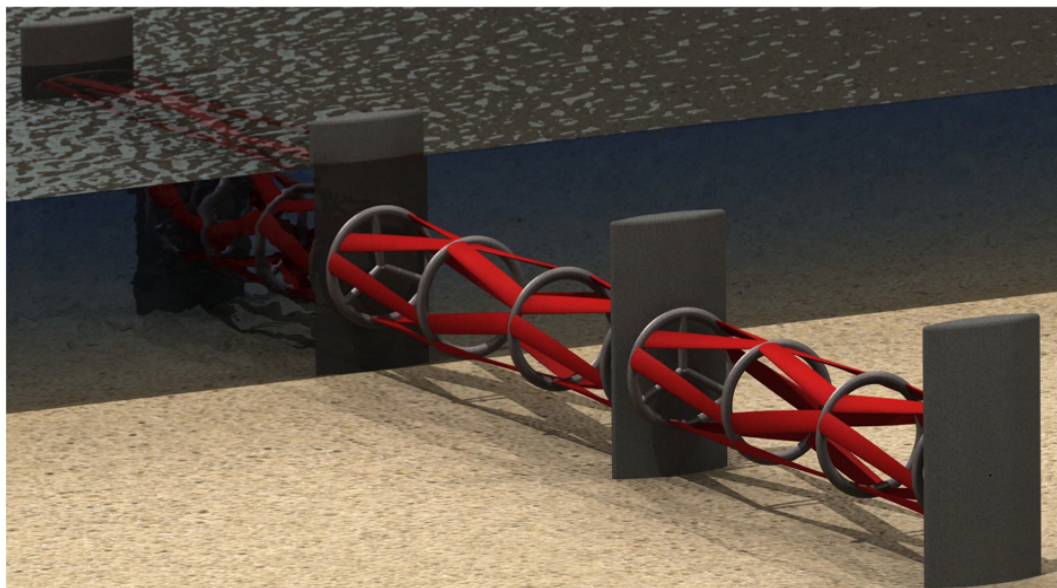


Figure 1.14: Artist's impression of an array of THAWTs, taken from McAdam et al. (2010)

1.4 Summary of research on cross-flow turbines

After the French engineer Darrieus (1931) had filed a patent for his his novel turbine design in 1926, for the following four decades little work was carried to improve the understanding of and further develop cross-flow turbines. It was only after the 1973 Arab oil embargo, which led to a significant increase in the oil price, that alternative and renewable energy sources, e.g. wind and tidal power, captured the interest of the public and research community. For instance, Fraenkel and Musgrove (1979) published a study on the potential of generating energy from tidal and river currents.

In the early 1980s a number of investigations were conducted, in particular at the National Research Council of Canada, to examine the potential of cross-flow turbines as hydraulic kinetic energy converters, see Swan and Farrell (1983), Faure (1984), Faure et al. (1986) and Takamatsu et al. (1985). From these tests, it was concluded that large ocean cross-flow turbines were feasible, but it was also pointed out that the maximum power coefficients obtained during these tests compared unfavourably with the performance of known axial-flow turbines. Thereupon, further research was undertaken to improve the performance of Darrieus-type machines. One particular aspect that was considered essential to increasing the turbines' power output was blade design. Gregorek (1983) and Migliore (1983) are examples of studies that focused on optimising and adapting the blades' foil section to the specific needs of cross-flow turbines.

Furthermore, at the Sandia National Laboratories in the US, Strickland et al. (1981) conducted towing tank tests and Sheldahl and Klimas (1981) wind tunnel experiments to further the understanding of vertical axis cross-flow turbines and to examine their potential as wind energy converters. However, at the end of the 1980s the oil price dropped significantly and consequently the interest in wind and in particular in tidal energy did so, too. Research activities on cross-flow turbines slowed down and the few projects that still were initiated almost exclusively focused on vertical axis wind

cross-flow machines.

For example, Kirke (1998) carried out a study of vertical axis cross-flow wind turbines and examined the use of variable pitch to avoid blade stall and maintain a favourable angle of attack throughout the entire cycle in order to enhance the performance Darrieus-type turbines. The results presented in Kirke and Lazauskas (1991) showed that the peak efficiency of cross-flow turbines using a variable pitch mechanism were up to 30% higher than that of a corresponding fixed pitch turbine. Also, in Paraschivoiu (2002) it is argued that vertical axis cross-flow wind turbines could have presented a competitive alternative to axial-flow machines, had they received the sufficient investment, which would have allowed for the necessary innovations. However, as apparent today, the large wind turbine developers almost exclusively focused on axial-flow turbines for large scale energy generation, because they operated with higher efficiencies than other turbine designs and were considered the most cost-effective solution for wind energy generation.

But over the last decade, cross-flow wind turbines have captured new interest by turbine developers as potential wind energy converters in urban surroundings. In Mertens et al. (2003), it was shown that the performance of a vertical axis cross-flow wind turbine was superior to the performance of the corresponding axial-flow machine, if operating under skewed flow conditions typical on flat roofs of high buildings. This is why a number of developers of small wind turbines for the urban/built environment, e.g. Turby from the Netherlands and Quiet Revolution from the UK, have decided to use cross-flow turbine designs; see Figure 1.15.



(a) Turby (2011)



(b) Quiet Revolution (2011)

Figure 1.15: Examples of cross-flow wind turbine concepts for an urban environment

But today cross-flow turbine designs are not only being considered for small-scale wind energy generation, but also by some members of the tidal stream turbine community. As discussed above, the majority of tidal stream turbine developers opt for axial-flow rotor designs, but a number of research projects have been conducted over the last few years to examine the feasibility of using cross-flow turbines as marine energy converters.

One of the key areas that has been re-investigated for tidal cross-flow turbines is the possibility of overcoming the cross-flow turbine's inherently lower efficiency relative to that of axial-flow devices, as discussed above, by using pitch control, for example see Gretton and Bruce (2005) and Hwang et al. (2009). In Salter and Taylor (2006), it has been suggested that the performance of marine cross-flow turbines with pitch control can be as good as for axial-flow machines. Moreover, the results from the numerical investigation presented in Gretton et al. (2009) show for a 3-bladed turbine with a turbine solidity of 0.048 that limiting the maximum angle of attack to 10° by pitch

control resulted in an increase in the maximum power take-off of about 7% in comparison to the fixed pitch variant. However, it is also discussed that it remains to be investigated how commercially viable a variable pitch mechanism is in a marine environment. One of the main advantages of fixed pitch cross-flow over axial-flow turbines is the simplicity of the design. Due to the inherent multi-directionality of a Darrieus-type turbine, cross-flow turbines do not require additional rotors or variable pitch mechanisms to accommodate for the bi-directionality of tidal flows, as many axial-flow machines do, e.g. the MCT or ARC turbines discussed in Section 1.3.1.1. Incorporating a variable pitch mechanism would put this advantage into question and may render cross-flow turbine concepts uneconomic relative to more conventional designs.

Also, the possibility of using a duct around a rotor of a tidal cross-flow turbine has been studied by Klaptocz et al. (2007), which showed a significant increase in the power produced by the ducted device relative to the bare cross-flow turbine. The advantages and disadvantages of ducted devices are outlined in Section 1.3.1.4.

Furthermore, a number of research projects have been carried out to examine the effect of changes in various design parameters on the performance of marine cross-flow turbines. For example, Shiono et al. (2000) conducted physical experiments to study how the performance of a marine cross-flow turbine varied with turbine solidity. The experiments showed that both the torque generated as well as the rotor's efficiency decreased with an increase in the number of blades due to a corresponding increase in the negative interference effects. In order to avoid significant variations in the torque produced with changes in the azimuthal position, it was concluded that a three-bladed turbine would be optimal.

A further interesting computational examination was carried out by Anthaume et al. (2008), who examined the dependency of turbine efficiency on single turbine vs. farm conditions. It is concluded that the efficiency of a single isolated (vertical-axis) cross-flow turbine is greatly increased when set both into a tower and into a cluster of several

towers corresponding to possible power farm arrangements. As discussed above, the increase in power output may be attributed to the increase in effective blockage due to the turbine farm arrangement.

So far, most studies on cross-flow turbines, both wind and tidal, have been conducted with vertical axis cross-flow turbines in mind. However, for the present study, we are interested in horizontal axis cross-flow turbines, which offer the possibility of forming long arrays stretched laterally across a tidal flow, as discussed in Section 1.3.2.1, and may thus lead to further increases in turbine performance than reported in Antheaume et al. (2008) associated with increasing the effective blockage.

McAdam et al. (2010) have conducted physical experiments of various configurations of a 1/20th scale horizontal axis cross-flow water turbine; for a detailed account of the experimental set-up and procedure see McAdam (2008). A comparison of a straight-bladed and 'truss' turbine, which is shown in Figure 1.14, revealed a reduction in performance of 3 - 11% of the latter, which is considered small, when taking the structural benefits of the truss structure into account. In a highly blocked configuration, (kinetic) power efficiencies of up to 160% were achieved. The turbine that exhibited the best performance was a straight 3-bladed variant, but it is also discussed that such a configuration may not be the most economical due to a significantly poorer structural performance relative to turbines with a higher solidity.

McAdam's experiments have shown that further research is required both in terms of the hydrodynamic performance, which is the focus of the present study, as well as the structural performance of a horizontal axis cross-flow water turbine in order to assess the optimal design parameters, such as turbine solidity.

Chapter 2

Numerical Methods

The objective of the present study is to further the understanding of the flow physics of a generic horizontal axis cross-flow turbine and to examine how changes of various design parameters affect the performance of such a turbine. The two options to explore the hydrodynamics of cross-flow turbines are through (i) physical experiments or (ii) numerical investigations.

Physical experiments, such as those conducted by McAdam et al. (2010), are essential to prove the feasibility of any (tidal) turbine design and can provide insights as to both the flow physics as well as design optimisations. However, such tests are time consuming, costly and in particular full-scale test are impossible to replicate under laboratory conditions, i.e. Froude and Reynolds number cannot be matched.

Moreover, work by Shi (2008) on Particle Image Velocimetry (PIV) suggests that an experimental flow visualisation model is not suitable to enable an extensive hydrodynamic analysis of a cross-flow water turbine. The information obtained from PIV using the accessible equipment is not detailed enough and moreover, the analysis would be limited to the configurations available for experiments.

Hence, for the present study it was determined to use a numerical model enabling a detailed analysis of the turbine's flow physics and performance. As discussed in

Section 1.3.1, much research has been carried out relating to the aerodynamics of wind turbines and below we consider whether some of the theories and modelling techniques developed for the wind industry may be useful for the present study.

2.1 Modelling techniques

2.1.1 Blade element momentum theory

In Section 1.3.1, the momentum actuator disc concept as well as the blade element theory were introduced. By combining these two theories, the induction factors and consequently the machine's power output can be determined. Blade element momentum theory (BEMT) typically relies on a model consisting of multiple concentric streamtubes in order to accommodate for the velocity variation across the rotor, for more detail see Strickland (1975); Figure 2.1 shows an example of such a model.

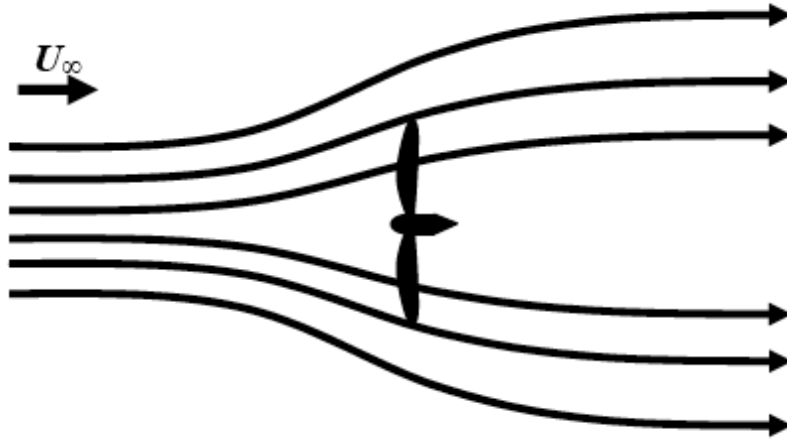


Figure 2.1: Illustration of multiple concentric streamtubes, taken from Gaden (2007)

As discussed in Burton et al. (2001), it is important to take note of the assumption of no radial interaction between the flows through contiguous annuli. It is assumed that it is only the force of the blade element that causes the change of momentum of the

fluid that passes through each streamtube.

By equating the respective expressions obtained by applying conservation of linear and angular momentum and by BET, and following an iterative procedure to calculate the axial and circumferential induction factors of the corresponding annular sections, the torque contribution of the rotor within each streamtube can be determined. Furthermore, by integrating the axial thrust and torque values of each annular section, i.e. through Equation 2.1.1 & 2.1.2, first discussed in Section 1.3.1.3, the rotor's overall performance, power, P , and thrust, T , may be determined.

$$T = B \int_{root}^{tip} (L \cos \phi + D \sin \phi) dr \quad (2.1.1)$$

$$Q_T = B \int_{root}^{tip} r (L \sin \phi - D \cos \phi) dr \quad (2.1.2)$$

$$P = Q_T \omega \quad (2.1.3)$$

where Q_T is the global torque, B the number of blades, L and D are the sectional lift and drag forces, r the radius, ϕ denotes the flow angle, and ω the angular velocity.

BEMT has proven a useful design tool in the wind and helicopter industries and has been further developed for the application to cross-flow turbines, see Paraschivoiu (1983) and Sharpe (1990). Flow through a cross-flow turbine passes the turbine twice, once on the upstream and once on the downstream passage, and thus a double actuator disc model has been developed comprising an upstream and downstream disc.

However, as discussed by Coiro et al. (2005), the convergence and accuracy of force predictions of BEMT models are typically adversely affected by high induction factors as well as the occurrence of dynamic stall, which are both possible for the turbines of interest for the present study. Moreover, potential effects due to flow confinement arising

from a turbine’s proximity to the free surface, whose importance has been discussed in Section 1.3.1, are not considered in (conventional) BEMT models.

In order to account for domain constraint effects, McAdam (2008) developed a numerical model that uses BET to provide the parameters required to solve the equations in the momentum actuator disc model for an open channel flow, derived by Houlby et al. (2008). A comparison of the results predicted by this model to those obtained from the physical experiments outlined in McAdam (2008) and McAdam et al. (2010) shows that the initial predictions by the numerical model significantly over-estimate the turbine’s power output, on average by more than 50%. Moreover, the numerical model predicted greater tip speed ratios for stall and peak power. Through a modification of the flow velocity on the downstream passage of the turbine, which seemed to have been over-predicted by the numerical model, the accuracy of the computations could be improved significantly. However, these adjustments were based on visual observations made during the physical experiments and McAdam (2008) was unable to improve the numerical model independently of experimental turbine performance data. Hence, McAdam (2008) concluded that a more appropriate technique should be developed for accurately predicting the performance of a cross-flow water turbine.

Furthermore, information about the blade forces and the rotor’s performance, as provided by BEMT models, is not sufficient for the analyses to be carried out for the present study. Computations of flow-field variables, such as the velocity and pressure distribution, are desired also.

2.1.2 Vortex models

Vortex models normally use singularity elements, e.g. vortex lines to model the blades of a turbine. These elements perturb the flow in a prescribed manner. Determining the elements’ strengths allows the computation of the flow field variables, from which the

blade lift and drag forces may be calculated using aerofoil section data, in turn allowing the computation of the rotor's performance.

For a Darrieus-type turbine Strickland et al. (1981) first developed a three-dimensional vortex model. Comparisons between the predicted blade forces using the vortex model to the corresponding results from physical experiments exhibited good agreement. Coiro et al. (2005) reported that results computed using BEMT and vortex models were similar, but that vortex models were more reliable for high induction factor simulations.

As for the BEMT model, vortex models rely on experimental or computed data for the lift and drag coefficients, C_L & C_D , as input variables, which is a drawback to their use for cross-flow turbine simulations. Firstly, the necessary data, particularly at low Reynolds number, is not readily available, so that extensive wind tunnel tests would have to be carried out or suitable approximations made. Secondly, and more importantly, it is uncertain how useful such C_L & C_D data would be, because the flow field, in which blades of cross-flow turbines operate, are so fundamentally different to that of axial-flow turbines or aircraft wings; for instance, the effect of flow curvature, see Migliore et al. (1980), arising from a cross-flow turbine blade's orbital motion, as discussed in more detail in 5.1.2, have not been fully understood. Moreover, C_L & C_D data from static blade tests for separated flow would introduce further uncertainties, for instance with regard to the occurrence and effects of dynamic stall. The complexity of the flow through a cross-flow water turbine necessitates a more complete model. Vortex models were hence considered unsuitable for the present study.

2.1.3 Computational Fluid Dynamics (CFD)

Hence, it was determined to use a Computational Fluid Dynamics (CFD) model to further the understanding of cross-flow turbines' hydrodynamics. In addition to providing the turbine's torque and power outputs, such a model allows the computation of the blade pressure distributions in addition to the entire velocity and pressure field, hence

enabling detailed analyses of flow patterns through and around the turbine as well as the effect of dynamic stall, flow interference etc.

As described in Anderson (1985), CFD is the art of replacing the integrals or partial derivatives in the equations that describe fluid problems with discretised algebraic forms, which, in turn, may be solved numerically to obtain numbers for the flow field values at discrete points in time and space. In fluid dynamics, there are three governing equations, which may be summarised as follows:

(i) Conservation of mass, which states that the mass of a closed system will remain constant over time; mass is neither created nor destroyed:

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \vec{U}) = 0 \quad (2.1.4)$$

where ρ is the fluid's density and $\vec{U}$ the velocity vector. For an incompressible fluid, Equation 2.1.4 simplifies to:

$$\nabla \cdot \vec{U} = 0 \quad (2.1.5)$$

(ii) Conservation of momentum (Newton's second law), which states that the rate of change of momentum of a system is equal to the sum of the forces on the system:

$$\rho \frac{D\vec{U}}{Dt} = \nabla \cdot \tau_{ij} + \rho \vec{g} + \vec{F} \quad (2.1.6)$$

where $\frac{D}{Dt}$ is the substantial derivative, defined in Equation 2.1.7, τ_{ij} is the stress tensor, $\rho \vec{g}$ and $\vec{F}$ the gravitational and external body forces per unit volume.

$$\frac{D}{Dt} = \frac{\partial}{\partial t} + (\vec{U} \cdot \nabla) \quad (2.1.7)$$

As discussed in Potter and Wiggert (2002), many fluids, e.g. water, oil and air, exhibit

a linear relationship between the stress components and the velocity gradients; these fluids are called Newtonian fluids. Moreover, if the fluid is isotropic, i.e. the fluid properties are independent of direction, the stress and velocity components can be related using only two fluid properties, the (dynamic) viscosity μ and the second coefficient of viscosity λ . The stress-velocity gradient relations, also called the constitutive relations, are given in Equation 2.1.8:

$$\tau_{ij} = -P\delta_{ij} + \mu \left(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right) + \delta_{ij}\lambda \frac{\partial U_i}{\partial x_i} \quad (2.1.8)$$

where P is the static pressure and δ_{ij} the Kronecker delta function; $\delta_{ij} = 1$, if $i = j$ and $\delta_{ij} = 0$ if $i \neq j$.

Now, when substituting Equation 2.1.8 into Equation 2.1.6, we obtain the Navier-Stokes equations for compressible viscous flows:

$$\rho \frac{D\vec{U}}{Dt} = -\nabla P + \frac{\partial}{\partial x_j} \left[\mu \left(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right) + \delta_{ij}\lambda \frac{\partial U_i}{\partial x_i} \right] + \rho \vec{g} + \vec{F} \quad (2.1.9)$$

If density and viscosity are constant, Equation 2.1.9 may be further simplified to:

$$\rho \frac{D\vec{U}}{Dt} = -\nabla P + \mu \nabla^2 \vec{U} + \rho \vec{g} + \vec{F} \quad (2.1.10)$$

where ∇^2 is the Laplacian defined as follows:

$$\nabla^2 = \frac{\partial^2}{\partial x^2} + \frac{\partial^2}{\partial y^2} + \frac{\partial^2}{\partial z^2} \quad (2.1.11)$$

(iii) Conservation of energy (first law of thermodynamics), which states that the total energy of an isolated system remains constant. The equation for energy conservation

only need be resolved for flows involving heat transfer or compressibility, which is not the case for the present study.

Hence, for incompressible fluids, Equation 2.1.5 and 2.1.10 are to be solved for velocity and pressure. However, for most engineering problems they cannot be solved analytically, so that approximate solutions to the governing equations need to be determined through CFD. This may be accomplished by replacing the continuous problem domain with a discrete domain using a grid and computing the relevant flow variables at the grid points, see Bhaskaran and Collins (2011); this is further discussed in Section 2.2.3.

2.2 Present numerical model - ANSYS FLUENT 12.0

We choose to use the commercial CFD software ANSYS FLUENT 12.0, see ANSYS Inc. (2009), which solves the Reynolds-Averaged Navier-Stokes (RANS) equations, discussed in Section 2.2.1, using a finite volume approach, see Section 2.2.3. For this study, ANSYS FLUENT is used as a two-dimensional, pressure-based, segregated, implicit, incompressible flow solver.

In the pressure-based approach the velocity field is computed from the momentum equations, while the pressure field is computed by solving a pressure correction equation, which is obtained by manipulating continuity and momentum equations; for detailed information see ANSYS Inc. (2009). The pressure (correction) equation is derived in such a manner, so that the velocity field, corrected by the pressure, satisfies the continuity equation.

The solver employed for this study used a segregated solution algorithm, in which the governing equations for the solution variables are solved sequentially from one another. When being solved, the individual governing equations are segregated from all other equations.

2.2.1 Turbulence - RANS equations

As discussed in Osalusi et al. (2009) and Gant and Stallard (2008), typical sites for tidal stream energy generation exhibit large scales of turbulent motion. Most tidal stream turbines will operate in highly turbulent flows and hence it is important that numerical tests of marine turbines are conducted under turbulent flow conditions.

As explicated in Bhaskaran and Collins (2011), turbulent flows are characterised by large, nearly random fluctuations in velocity and pressure in both space and time. The fluctuations arise from instabilities that grow until nonlinear interactions lead them to decompose into smaller and smaller eddies, which are finally converted into heat

through viscous dissipation. With regard to modelling turbulent flows, it is important to note that the fluctuations discussed above result in mixing of transported quantities, such as momentum, and the transported quantities fluctuate, too. For most engineering applications, these fluctuations are too computationally expensive to simulate directly, as they can be of small scale and high frequency. In response, the so-called Reynolds-averaged Navier-Stokes (RANS) equations, see Equation 2.2.3 & 2.2.4, have been derived, which govern the transport of the time-averaged flow quantities. These equations are computationally significantly less expensive to solve, but yield six additional unknown variables, which require supplementary turbulence models to “close” the system of equations.

In Reynolds averaging, the fully resolved fluctuating flow variable, ϕ , is decomposed into a steady mean component, $\bar{\phi}$, and a time varying fluctuating component, ϕ' , which itself has a zero mean value, see Versteeg and Malalasekera (2007):

$$\phi(t) = \bar{\phi} + \phi'(t) \quad (2.2.1)$$

where

$$\bar{\phi} = \frac{1}{\Delta t} \int_0^{\Delta t} \phi(t) dt \quad (2.2.2)$$

where Δt is a long time scale in relation to that of the turbulent fluctuations.

Substituting expressions of this form for the flow variables into the continuity and momentum equations, see Equation 2.1.5 & 2.1.10, and taking a time average yields the so-called Reynolds-averaged Navier-Stokes (RANS) equations:

$$\nabla \cdot \vec{U} = 0 \quad (2.2.3)$$

$$\rho \frac{D\vec{U}}{Dt} = -\nabla P + \mu \nabla^2 \vec{U} - \rho \frac{\partial}{\partial x_j} (\overline{u'_i u'_j}) + \rho \vec{g} + \vec{F} \quad (2.2.4)$$

where the overbar on the mean velocity and static pressure has been dropped, such that $\vec{U}$ corresponds to the time-mean velocity and P to the time-mean static pressure; u'_i denotes the fluctuating velocity component, and the overbar represents a time-average.

The structure of the RANS equations is similar to that of the instantaneous Navier-Stokes equations, but the velocities and other solution variables in the RANS equations are time-averaged values. Moreover, the time-averaged momentum equation includes an additional term to represent the effects of turbulence, the Reynolds stresses, $-\rho \overline{u'_i u'_j}$; this additional term introduces six unknowns, which need to be modelled in order to close the RANS equations.

2.2.2 Turbulence models

2.2.2.1 The Boussinesq approximation

The Boussinesq hypothesis, see Boussinesq (1877), allows to compute the Reynolds stresses by relating them to the mean velocity gradients as follows:

$$-\rho \overline{u'_i u'_j} = \mu_t \left(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right) - \frac{2}{3} \rho k \delta_{ij} \quad (2.2.5)$$

where the turbulent kinetic energy, $k = \frac{1}{2} (\overline{u'^2} + \overline{v'^2} + \overline{w'^2})$; μ_t is the turbulent viscosity and typically calculated as a function of one or two transport variables, as further discussed below. In fact, when applying the Boussinesq approximation, the closure

problem becomes that of determining the turbulent viscosity, μ_t , and turbulent kinetic energy, k .

It is important to note that the Boussinesq hypothesis assumes that the turbulent viscosity, μ_t , is isotropic, i.e. the same in all directions, which is not the case for all flows.

A large number of different turbulence models employing the Boussinesq approach have been developed. Typically, these turbulence models have been classified according to the number of additional transport equations they use to compute the turbulent viscosity, μ_t .

2.2.2.2 Zero-equation models

Zero-equation models do not require, as the name suggests, the solution of any additional (transport) equations. The turbulent viscosity is calculated from existing quantities, e.g. mean flow velocities. Zero-equation models are easy and cheap to implement and are typically used for computing simple flows, such as jets or mixing layers, as discussed in Versteeg and Malalasekera (2007). However, predictions for separated flows are generally poor with zero-equation models, as explained in Wilcox (1994), and are hence considered inappropriate for the cross-flow turbine simulations of the present study. Examples of zero-equation models are the Cebeci-Smith and Baldwin-Lomax models, for more information see Wilcox (1994).

2.2.2.3 One-equation models

One-equation models typically comprise one transport equation for the turbulent viscosity and thus account for the flow history, hence providing a more physically realistic model in contrast to zero-equation models.

However, the turbulent length scale is not computed, which can render one-equation

models insensitive to transport processes. As discussed in ANSYS Inc. (2009), one-equation models have been criticised for not being able to accommodate changes in length scale, as might be necessary when the flow changes abruptly from a wall-bounded to a free shear flow. Instead, the turbulence length scale is related to some typical flow dimension, which is often difficult to define in complex geometries.

An example of a one-equation model is the Spalart-Allmaras model, which is discussed in more detail in Section 2.2.2.5.

2.2.2.4 Two-equation models

Two-equation models are probably the most widely used type of turbulence model. In Durbin (1995) and Vaz et al. (2007), it is discussed that (at least) a two-equation model is required to model complicated flows, such as massively separated boundary layers.

Two-equation models comprise two additional transport equations to represent the turbulent properties of the flow and are thus more computationally expensive than one-equation models. But the solution of two separate transport equations allows turbulent velocity and length scales to be calculated independently; such models also account for history effects such as convection and diffusion of turbulent energy.

Typically, the turbulence kinetic energy, k , is one of the transported variables and common choices for the second variable are the dissipation rate of the turbulence kinetic energy, ϵ , or the specific dissipation rate, ω . The first variable, k , represents the energy in the turbulence and the second variable is used to model the (length or time) scale of the turbulence. In Section 2.2.2.6, an example of a two-equation model, SST $k - \omega$, is described in more detail.

As mentioned above, a large number of different turbulence models are available. In addition to those using the Boussinesq approximation, other models, for instance the

Reynolds Stress Transport Models exist, which solve transport equations for each of the terms in the Reynolds stress tensor, resulting in additional computational expense; for more detail see Launder (1975).

After an investigation of the turbulence models used for numerical studies with similar requirements to those of the present study and after completing a literature research of the advantages and disadvantages of the various models, the author decided to further examine and test the feasibility of a one-equation and a two-equation model for the simulations of the present study:

- the Spalart-Allmaras model, which was also used for simulating flows through marine cross-flow turbines by Uglow (2008);
- the SST $k - \omega$ model, which was employed for simulating flows through marine cross-flow turbines by Grettton et al. (2009) and Dai and Lam (2009).

Both models are discussed in more detail below and were used and tested during the validation of the present numerical model, which is presented in Chapter 3.

2.2.2.5 Spalart-Allmaras (S-A)

Spalart and Allmaras (1992) originally developed their model for aerospace applications involving wall-bounded flows and has been found to perform well for boundary layer flows subjected to adverse pressure gradients.

The Spalart-Allmaras (S-A) turbulence model adopts the Boussinesq approach, outlined in Section 2.2.2.1. The model comprises a transport equation for the variable $\tilde{\nu}$, which is identical to the turbulent kinematic viscosity, ν_t , apart from in the near-wall region. The conservation equation for $\tilde{\nu}$ is:

$$\frac{\partial (\rho \tilde{\nu})}{\partial t} + \nabla \cdot (\rho \tilde{\nu} \vec{U}) = \frac{1}{\sigma_{\tilde{\nu}}} \{ \nabla \cdot [(\mu + \rho \tilde{\nu}) \nabla \tilde{\nu} + C_{b2} \rho] + (\nabla \tilde{\nu})^2 \} + G_{\nu} - Y_{\nu} \quad (2.2.6)$$

where $\sigma_{\tilde{\nu}}$ and C_{b2} are constants, G_{ν} is a source term representing the production of turbulent viscosity and Y_{ν} the destruction of turbulent viscosity that occurs in the near-wall region due to wall blocking and viscous damping.

The production and destruction terms are closed by a system of equations that introduce multiple empirically evaluated coefficients, for detailed information see ANSYS Inc. (2009).

The turbulent viscosity, μ_t , is computed as follows:

$$\mu_t = \rho \tilde{\nu} f_{v1} \quad f_{v1} = \frac{\chi^3}{\chi^3 + C_{v1}^3} \quad \chi = \frac{\tilde{\nu}}{\nu} \quad (2.2.7)$$

where f_{v1} is a viscous damping function and C_{v1} a constant.

The ANSYS FLUENT 12.0 implementation of the S-A turbulence model, see ANSYS Inc. (2009), uses the following model coefficient values:

$$C_{b2} = 0.622 \quad \sigma_{\tilde{\nu}} = \frac{2}{3} \quad C_{v1} = 7.1 \quad (2.2.8)$$

Also, it is noted that the last term in Equation 2.2.5 is ignored when calculating the Reynolds stresses, as the turbulence kinetic energy, k , is not computed in the S-A model.

2.2.2.6 Shear Stress Transport (SST) $k - \omega$

The Shear Stress Transport model (SST) $k - \omega$ was developed by Menter (1994) with the intention to combine the respective strengths of the standard $k - \omega$ and $k - \epsilon$ turbulence models in one model; the former provides a robust and accurate formulation for the near-wall region, whilst the latter offers free-stream independence in the far-field, as discussed in ANSYS Inc. (2009). The SST $k - \omega$ model uses a blending function, which activates the standard $k - \omega$ model for the near wall region, whilst the $k - \epsilon$ model is activated away from the surface.

For simulating external aerodynamic flows, Versteeg and Malalasekera (2007) conclude that the SST $k - \omega$ turbulence model is more general than the S-A and standard $k - \omega$ models and exhibits superior performance for adverse pressure gradient boundary layers as well as free shear layers. Also, in ANSYS Inc. (2009), it is suggested that the SST $k - \omega$ model is more accurate and reliable for a wide class of flows, including aerofoils, than other two-equation models.

The SST $k - \omega$ model also belongs to the group of turbulence models that employ the Boussinesq approximation, outlined in Section 2.2.2.1. The SST $k - \omega$ model solves two transport equations, one for the turbulence kinetic energy, k , and the other for the specific turbulent dissipation rate, ω :

$$\frac{\partial (\rho k)}{\partial t} + \nabla \cdot (\rho k \vec{U}) = \nabla \cdot (\Gamma_k \nabla k) + \tilde{G}_k - Y_k \quad (2.2.9)$$

$$\frac{\partial (\rho \omega)}{\partial t} + \nabla \cdot (\rho \omega \vec{U}) = \nabla \cdot (\Gamma_\omega \nabla \omega) + G_\omega - Y_\omega + D_\omega \quad (2.2.10)$$

where $\tilde{G}_k$ and G_ω represent the production of k and ω respectively, Y_k and Y_ω the

dissipation of k and ω respectively; D_ω is the cross-diffusion term and Γ_k and Γ_ω are the effective diffusivity of k and ω respectively.

The production, dissipation, cross-diffusion terms and the effective diffusivity are closed by a system of equations that introduce multiple empirically evaluated coefficients, for detailed information see ANSYS Inc. (2009).

The turbulent viscosity, μ_t , is computed as follows:

$$\mu_t = \frac{\rho k}{\omega} \frac{1}{\max \left[\frac{1}{\alpha^*}, \frac{SF_2}{\alpha_1 \omega} \right]} \quad (2.2.11)$$

where S is the strain rate magnitude, see Equation 2.2.12, F_2 a blending function, see Equation 2.2.13, a_1 a constant and α^* is given by Equation 2.2.14:

$$S \equiv \sqrt{2\Omega_{ij}\Omega_{ij}} \quad \Omega_{ij} = \frac{1}{2} \left(\frac{\partial U_i}{\partial x_j} - \frac{\partial U_j}{\partial x_i} \right) \quad (2.2.12)$$

$$F_2 = \tanh(\Phi_2^4) \quad \Phi_2 = \max \left[2 \frac{\sqrt{k}}{0.09\omega y}, \frac{500\mu}{\rho y^2 \omega} \right] \quad (2.2.13)$$

$$\alpha^* = \alpha_\infty^* \left(\frac{\alpha_0^* + Re_t/Re_k}{1 + Re_t/Re_k} \right) \quad Re_t = \frac{\rho k}{\mu \omega} \quad (2.2.14)$$

where y is the distance to the next surface and Re_k , α_0^* and α_∞^* are constants.

The ANSYS FLUENT 12.0 implementation of the SST $k - \omega$ turbulence model, see ANSYS Inc. (2009), uses the following model coefficient values:

$$a_1 = 0.31 \quad Re_k = 6 \quad \alpha_0^* = 0.024 \quad \alpha_\infty^* = 1 \quad (2.2.15)$$

2.2.3 Spatial discretisation: Finite Volume Method

As mentioned in Section 2.1.3 and discussed in detail in Bhaskaran and Collins (2011), the strategy of CFD consists of replacing a continuous problem domain with a discrete domain and solving for the relevant flow variables at the resulting grid points. This is further illustrated by Figure 2.2.

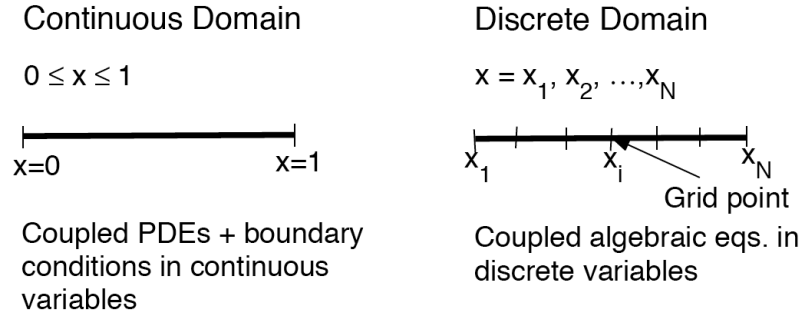


Figure 2.2: Continuous vs. discrete domain, taken from Bhaskaran and Collins (2011)

While a flow variable is defined at every point in a continuous domain, it is (only) defined at the grid points in a discrete domain, see Figure 2.2. For example, in a one-dimensional domain (1D), the pressure is defined in a continuous domain, as shown in Equation 2.2.16, but in a discrete domain as shown in Equation 2.2.17:

$$P = P(x), \quad 0 < x < 1 \quad (2.2.16)$$

$$P_i = P(x_i), \quad i = 1, 2, \dots, N \quad (2.2.17)$$

With regard to the governing partial differential equations outlined above, it is important to note that they were derived in terms of continuous variables, such as $P, \vec{U}$. For

the discrete domain, they need to be approximated using the corresponding discrete variables, e.g. $P_i, \vec{U}_i$. This process is performed in ANSYS FLUENT using the finite volume method (FVM).

The FVM may be summarised as follows, see for example Versteeg and Malalasekera (2007) or Ferziger and Peric (2002) for further details:

1. Grid generation: The computational domain is split into a number of control volumes, typically called cells or elements, and the variables are evaluated at the centroid of the control volume, for example see the triangular cell in Figure 2.4.
2. Discretisation: The differential form of the governing equations, see Equation 2.2.3 and 2.2.4, is integrated over each control volume. The variation of the variables between cell centroids are approximated using interpolation profiles.
3. Solution of equations: The resulting discretised/algebraic equation, which represents the conservation principle for the variable inside the control volume, can be solved numerically.

Below an illustration of the discretisation of the continuity equation is given. As discussed in Bhaskaran and Collins (2011), for steady, incompressible flow applying Green's theorem, see Kaplan (1991), to the integral form of the continuity equation renders:

$$\int_S \vec{U} \cdot \hat{n} dS = 0 \quad (2.2.18)$$

where the integration is carried out over the surface S of the control volume and $\hat{n}$ is the outward normal at the surface.

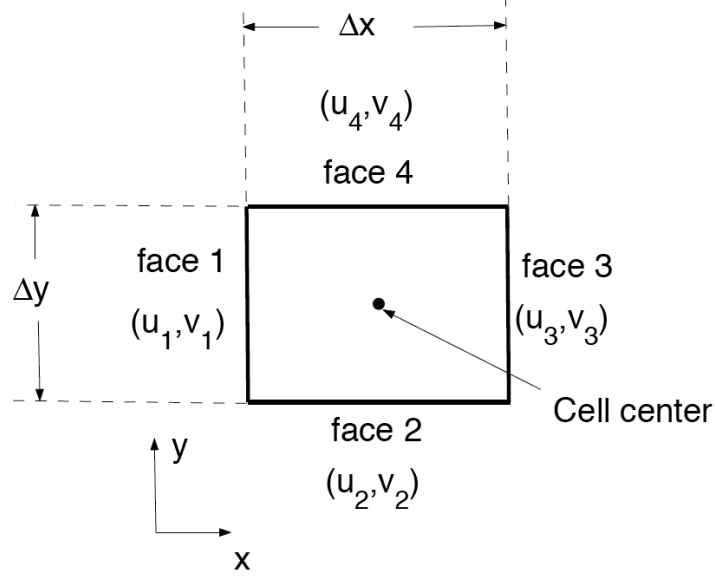


Figure 2.3: Rectangular cell, taken from Bhaskaran and Collins (2011)

A semi-discrete version can be derived by replacing the line integral with a summation over the discretised boundary of the control volume, which is defined by the rectangular cell shown in Figure 2.3, resulting in the following (discrete) equation:

$$-u_1\Delta y - v_2\Delta x + u_3\Delta y + v_4\Delta x = 0 \quad (2.2.19)$$

where the velocity vector at face i is assumed as $\vec{U}_i = u_i\hat{i} + v_i\hat{j}$.

Equation 2.2.19 states that net mass flow into the cell is zero, i.e. conservation of mass. Similarly, discrete equations may be derived for the conservation of momentum for each cell.

The general approach used in the present finite volume implementation is to convert the scalar transport equations to algebraic equations, further details may be found in ANSYS Inc. (2009). The triangular cell in Figure 2.4 is an example of a control volume, V , to which the integral form of the unsteady conservation equation for transport of a scalar quantity ϕ , as given in Equation 2.2.20, is applied:

$$\int_V \frac{\partial \rho \phi}{\partial t} dV + \oint \rho \phi \vec{U} \cdot d\vec{A} = \oint \Gamma_\phi \nabla \phi \cdot d\vec{A} + \int_V S_\phi dV \quad (2.2.20)$$

where $\vec{A}$ is the surface area vector, Γ_ϕ the diffusion coefficient for ϕ and S_ϕ the source of ϕ per unit volume.

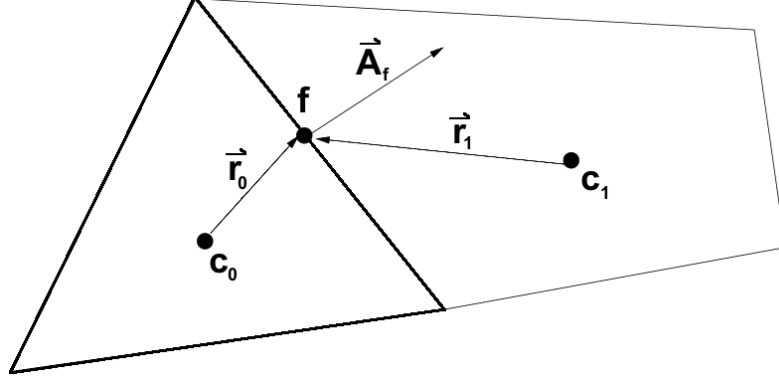


Figure 2.4: Control volume (bold edge) used to illustrate discretisation of a scalar transport equation, adapted from ANSYS Inc. (2009)

Equation 2.2.20 is applied to each cell in the computational domain. One of the key advantages of the FVM is its suitability for different kinds of meshes, e.g. multi-dimensional, structured as well as unstructured. The meshes employed for the present study were generated with the commercial software GAMBIT 2.1, see FLUENT Inc. (2003). For specific information on the various meshes used in the present study see the chapters on validation and results.

In the simplest implementation of the FVM all variables are approximated as piecewise constant taking the value at the cell centroid across the entire element. Following this approximation enables Equation 2.2.20 to be approximated for a given cell, yielding:

$$\frac{\partial \rho \phi}{\partial t} V + \sum_f^{N_{faces}} \rho_f \vec{U}_f \phi_f \cdot \vec{A}_f = \sum_f^{N_{faces}} \Gamma_\phi \nabla \phi_f \cdot d\vec{A} + S_\phi V \quad (2.2.21)$$

where N_{faces} is the number of faces enclosing the cell, ϕ_f the value of ϕ convected

through face f , $\rho_f \vec{U}_f \cdot \vec{A}_f$ the mass flux through the face, $\vec{A}_f$ the area of face f and V the cell volume; $\frac{\partial \rho \phi}{\partial t}$ is further discussed under temporal discretisation in Section 2.2.4.

As discussed above, the discrete values of the scalar ϕ are determined at the cell centres, for example c_0 and c_1 in Figure 2.4. The face values ϕ_f , which are required to evaluate fluxes in Equation 2.2.21, are interpolated from the cell centre values using an upwind scheme. As described in ANSYS Inc. (2009), upwinding means that the face value ϕ_f is derived from quantities in the cell upstream, or “upwind,” relative to the direction of the face normal velocity U_n .

2.2.4 Temporal discretisation

As the flow problems examined for the present study are unsteady, the evolution of the flow field is solved on a time marching basis. Consequently, the governing equations are to be discretised not only in space, but time as well. Temporal discretisation consists of integrating every term in the differential equations over a time step Δt . The method with which the integration of the transient terms is performed in ANSYS FLUENT is explicated below, for further details see ANSYS Inc. (2009).

A generic expression for the time evolution of a variable ϕ is represented by:

$$\frac{\partial \phi}{\partial t} = F(\phi) \quad (2.2.22)$$

where the function F includes any spatial discretisation. The second-order discretisation used in ANSYS FLUENT for the time derivative in Equation 2.2.22 is given by:

$$\frac{3\phi^{n+1} - 4\phi^n + \phi^{n-1}}{2\Delta t} = F(\phi^{n+1}) \quad (2.2.23)$$

where $n + 1$ is the value at the next time level, $t + \Delta t$, n the value at the current time level, t , and $n - 1$ the value at the previous time level, $t - \Delta t$. Note that $F(\phi)$ is

evaluated at time $n + 1$ indicating that the scheme is temporally implicit.

Equation 2.2.23 is solved iteratively at each time level before progressing to the next time step.

The choice of time step, Δt , is crucial with regard to the stability, convergence and computational cost of the simulations. The appropriate choice of time step depends upon the stability of the simulation, the scaled residuals reached, the number of iterations per time step required to attain the desired residual levels as well as the sensitivity of the solution to further reductions in Δt . ANSYS FLUENT computes residuals for the continuity and momentum equations, turbulence models as well as the volume of fluid technique, if applied.

An extensive study of solution convergence was conducted for the various turbines simulated under different flow conditions, outlined in Chapter 4 and 5. It was found that a scaled residual target of 1×10^{-4} to 1×10^{-5} , depending on turbine configuration, was sufficient to attain a converged solution; a solution was considered converged in terms of the computed residuals, if the power coefficient did not change to the second significant figure after further increasing the residual target. The time step used to attain the required residual levels, typically within less than 20 iterations per time step, varied from 5×10^{-3} seconds to 6×10^{-4} seconds depending on turbine configuration and Reynolds number.

The simulations were run on a cluster having 8 nodes with 2 quad-core 3.0 GHz CPUs per node. For the simulations discussed in Chapter 4 and 5 the core hour computation time per turbine revolution varied from 80 to 290 core hours, where core hours = wall clock hours $\times$ number of cores used; so for example, the simulation for $b = 50\%$ and $\lambda = 3$ discussed in Section 5.1.1 was considered statistically converged after 21 revolutions, where the computation time was 90 core hours per revolution, thus 1890 core hours in total.

2.2.5 Free surface model - Volume of Fluid (VOF)

One of the objectives of the present study is to examine potential effects of free surface deformation on the performance of a horizontal axis cross-flow tidal turbine. As discussed in Myers and Bahaj (2006) and Bryden et al. (2007), extracting energy from an open channel flow results in a drop in the free surface level. Hence, when investigating effects of free surface proximity, it is important to not only study flow confinement effects (with solid boundaries), but also to model the free surface deformation and analyse its effect on turbine performance.

To this end, a Volume of Fluid (VOF) model has been deployed for the free surface simulations presented in Chapter 5. In the VOF formulation the air-water interface is tracked by solving a momentum equation for the volume fraction of one (or more) of the phases/fluids. For the q^{th} phase, this equation is given as follows:

$$\frac{1}{\rho_q} \left[\frac{\partial}{\partial t} (\alpha_q \rho_q) + \nabla \cdot (\alpha_q \rho_q \vec{U}_q) \right] = \sum_{p=1}^n (\dot{m}_{pq} - \dot{m}_{qp}) \quad (2.2.24)$$

where α_q and α_p are the q^{th} and p^{th} volume fraction in a cell respectively, ρ_q and ρ_p the densities of phase q and p respectively, $\dot{m}_{qp}$ is the mass transfer from phase q to phase p and $\dot{m}_{pq}$ is the mass transfer from phase p to phase q .

For the present study, there are only two phases/fluids; air, defined as the primary (1), and water, defined as the secondary phase (2).

The volume fraction equation is only solved for the secondary phase, water; the volume fraction of air is computed as follows:

$$\alpha_1 = 1 - \alpha_2 \quad (2.2.25)$$

Fluid properties for cells containing both air and water are computed in the manner shown below for density, ρ :

$$\rho = \sum \alpha_q \rho_q \quad (2.2.26)$$

$$\rho = \rho_2 \alpha_2 + (1 - \alpha_2) \rho_1 \quad (2.2.27)$$

With regard to the boundary conditions employed in the present VOF model, the inflow conditions, both the water depth and flow velocity far upstream of the turbine at the inlet, are known, but the downstream conditions at the outlet are unknown, as they depend on turbine performance and energy extraction, which are to be computed via the simulations. By specifying both the inlet conditions and the outlet water depth, the energy removed from the domain is artificially fixed. Moreover, VOF simulations including rotors extracting energy and where the outlet water depth was fixed have shown (non-physical) free surface features arising from a difference in the net energy extracted from the domain and the energy extraction represented by the (pre-defined) deformation of the free-surface. The solution to this problem is to use an iterative technique that recursively adjusts the outlet flow depth to achieve the desired inflow conditions, with energy removal within the domain due to the turbine and mixing processes, and without spurious energy sinks/sources to account for the mismatch between boundary conditions and energy removal within the domain.

McIntosh et al. (2010) developed a User-Defined Function, which allows to balance the energy extracted within the domain to that reflected along the solution boundaries. The modelling strategy is summarised in McIntosh et al. (2010) as follows:

1. upstream water depth h_{in} is evaluated a short distance downstream of the inlet via a linear interpolation between phases;
2. upstream depth h_{in} is compared to the desired (or target) inlet height h_{in_target} to calculate a height error, $h_E = h_{in_target} - h_{in}$;

3. height error is multiplied by a gain k and the current time step size, Δt , and used to set a new outlet height $h_{out} = h_{out_old} - k\Delta t h_E$.

Note that the gain in the outlet adjustment is $-k$ such that a positive upstream perturbation results in a reduction in the outflow height, so that this upstream perturbation can be washed downstream and through the outflow boundary.

This method has been successfully validated in McIntosh et al. (2010) and was first implemented for computations of a cross-flow turbine for the simulations discussed in Section 5.2.

Adjustments of the gain factor k as well as a damping parameter, which controls the adjustment of the inlet velocity used to reduce the free surface oscillations near the inlet, allows the desired inflow conditions to be typically attained in less than 25 wave times for the cross-flow turbine simulations, where wave time, t_{wave} , is the time it takes for a wave to pass through the computational domain in the streamwise direction.

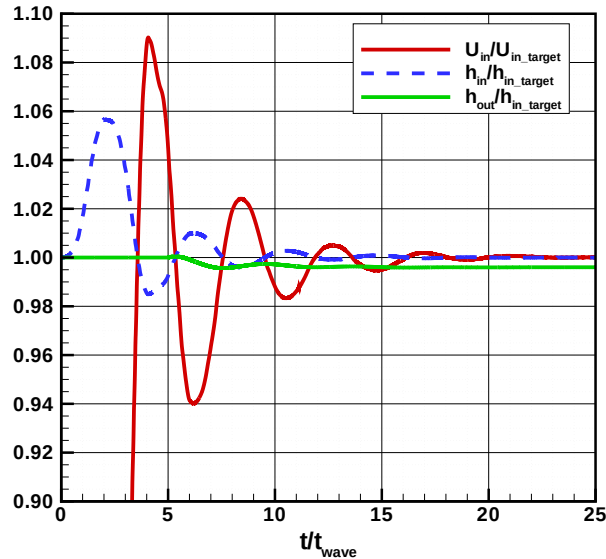


Figure 2.5: Time history of inlet and outlet water depth as well as inlet velocity

Figure 2.5 shows an example of how the inlet velocity and water height as well as the outlet water height vary with time for a cross-flow turbine simulation using the UDF described above to model the free surface. It may be observed that the desired inflow conditions are attained after about 25 wave times, whilst the outlet water depth reached its (final) height after about 15 wave times. Also, it is to be noted that the inflow velocity was ramped up from $U = 0 \text{ m/s}$ at $t = 0 \text{ s}$ in order to increase solution stability.

Chapter 3

Validation

The validation of the numerical model consisted of three stages:

1. static blade tests,
2. dynamic blade tests and
3. rotating turbine tests.

The static and dynamic blade tests were used to investigate the influence of numerical parameters such as mesh resolution, model discretisation and turbulence model; the oscillating blade tests allowed for the study of the dynamics of the blades' motion. The rotating turbine tests were required to validate the computation of characteristics specific to turbines, e.g. the influence of flow impedance, as well as characteristics specific to cross-flow turbines, e.g. blade-vortex interaction.

In the present investigation we are interested in high laboratory to field-test Reynolds numbers. Due to the availability of the respective data, static blade tests were carried out at blade Reynolds numbers, $Re_c = O(10^5)$ & (10^6) , the dynamic tests at $Re_c = O(10^6)$ and the rotating turbine tests at $Re_c = O(10^4)$, where:

$$Re_c = \frac{cU_\theta}{\nu} \quad (3.0.1)$$

where U_θ , c and ν are the incident flow speed, blade chord and kinematic viscosity respectively (in the case of a rotating turbine this is the mean blade Reynolds number around the azimuth and U_θ is the blade rotational velocity).

For comparison, full scale tidal turbines will operate at $Re_c = O(10^6) - O(10^7)$.

In the following sections, the various validation stages are discussed.

3.1 Reynolds number = $O(10^4)$ - $O(10^5)$

3.1.1 Static blade tests

The static blade tests consisted of comparing lift and drag data for a two-dimensional (2D) NACA 0015 aerofoil section over a range of incidences from wind-tunnel tests conducted at the Sandia National Laboratories (SNL), USA, see Sheldahl and Klimas (1981), to the corresponding results from the present numerical investigation. This section of the validation stage was carried out at $Re_c = 3.6 \times 10^5$.

In Section 3.1.1.1, the flow physics of the static blade tests are discussed using the results from the converged numerical solution, while the sensitivity of the present numerical solution to mesh refinement and turbulence models is discussed in Section 3.1.1.2.

3.1.1.1 Comparison of numerically and experimentally obtained lift and drag data

Figure 3.1 shows the lift and drag coefficients, C_L & C_D , from the present simulations, using the SST $k - \omega$ turbulence model, as well as the experimental data from Sheldahl and Klimas (1981), where

$$C_L = \frac{L}{\frac{1}{2}\rho U_\infty^2 c} \quad (3.1.1)$$

$$C_D = \frac{D}{\frac{1}{2}\rho U_\infty^2 c} \quad (3.1.2)$$

where L and D are the lift and drag per unit span and U_∞ the free-stream velocity.

For the physical experiments described in Sheldahl and Klimas (1981), the section coefficients of lift and drag were determined using a balance system. The results were corrected for wind tunnel wall effects as well as wind-turbulence factors, used to correct the test Reynolds number. Also, the static tests were conducted for both positive and negative incidence and differences of up to 40% in C_D were recorded between measurements for the positive and the corresponding negative angle of attack. The present author hence decided to use the mean value of the readings for the positive and respective negative angle of incidence for the SNL drag data plotted in Figure 3.1b.

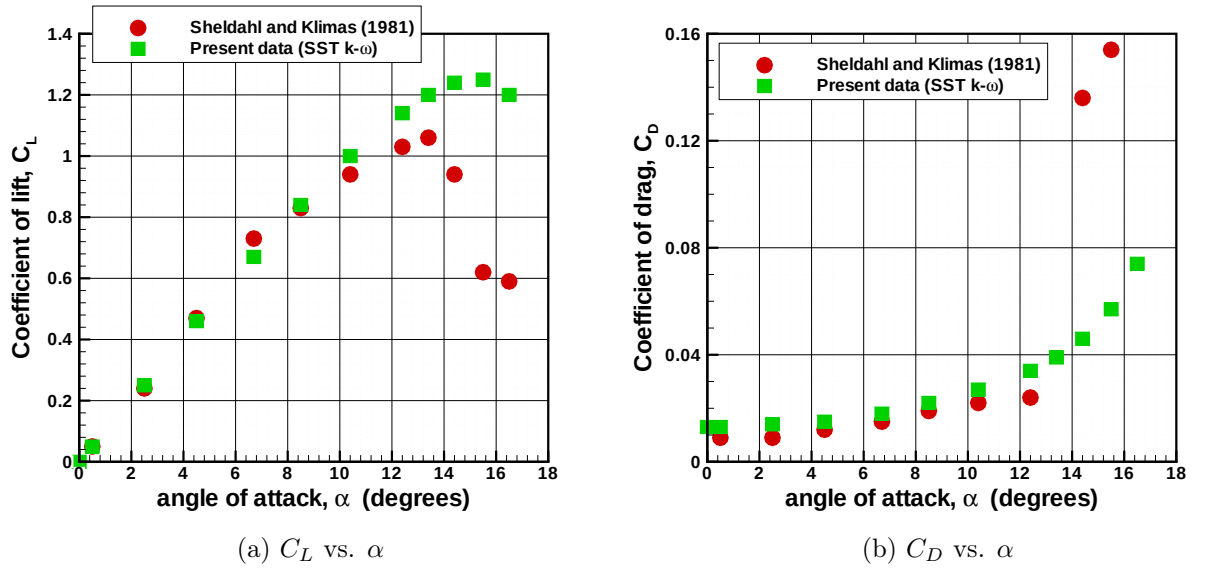


Figure 3.1: NACA 0015 blade at $Re_c = 3.6 \times 10^5$: comparison of numerically and experimentally obtained lift and drag coefficients

The experimentally derived lift and drag forces show the characteristics to be expected for a hydrofoil operating at a moderate Re . At incidences below the static stalling angle, α_{ss} , the coefficient of lift, C_L , increases approximately linearly with α and the coefficient of drag, C_D , is small in comparison to its post stall value. C_L reaches its maximum at around the static stalling angle, α_{ss} , while C_D increases significantly at around the same incidence due to the increase in pressure drag caused by flow separation. Moreover, the SNL tests exhibit the anticipated stalling mechanism, described as follows.

At moderate Reynolds numbers, Re , flow separation can occur well below α_{ss} , as discussed in Mayda and van Dam (2002). At low Re , where viscous forces can dampen disturbances that initiate the transition to turbulence, the boundary layer over a hydrofoil may be unable to make a natural transition from laminar to turbulent flow and will separate before becoming turbulent. Once separated, however, the separated shear layer may become unstable and undergo transition to turbulence, following which it may reattach to the blade's surface, hence forming a so-called separation bubble. Such bubbles are short at low incidence, but as α increases, the reattachment point moves further downstream and thus the bubble size increases. The formation of such a bubble leads to a sharp increase in drag, whilst its growth rate and hence its effect on lift depend on the Reynolds number, see Sharpe (1990). At moderate Reynolds numbers, $10^5 < Re < 2 \times 10^6$, the bubble typically spreads suddenly resulting in a sharp drop in lift, as shown in Figure 3.1 for the experimental SNL data.

When examining the results of the numerical simulations, two key differences to the experimental data are evident.

1. C_D for the attached flow region is significantly over-predicted, as shown in Table 3.1;

2. the transition from the attached to the stalled region is poorly simulated, see Figure 3.1; the peak in $\max C_L$ is over-predicted by about 2 degrees and $\max C_L$ by around 20%.

α	Error in C_D
0.5°	47%
2.5°	44%
4.5°	31%
6.7°	19%
8.5°	17%

Table 3.1: Percentage error in C_D pre-stall conditions

These deficiencies are due to the difficulty in modelling laminar to turbulent transition using a RANS approach. The few models that can accurately compute transition are not suitable for use in a dynamic flow environment such as that experienced by a cross-flow turbine, whose blades see a continually changing incidence. Existing transition models, for example see Langtry and Menter (2009), are of little use for a dynamic simulation and hence the CFD simulations presented here have been computed under fully turbulent flow conditions. Stipulating the boundary layer to be fully turbulent leads to an over-estimation of the skin friction upstream of where transition would occur in the real flow and hence an over-prediction of C_D for the attached flow region. As α increases, the real flow transition point moves forward and hence the error in the blade integrated skin friction and thus in blade drag decreases, as shown in Table 3.1.

Furthermore, the inability to simulate laminar to turbulent transition is responsible for the differences between the numerical and experimental results of the predictions of stall. Since a fully turbulent boundary layer was assumed throughout, the numerical simulations exhibit a fundamentally different stalling mechanism, which is more like that seen at higher Re . As is expected at higher Re , the flow separated from the trailing edge and, as α increased, the point of separation moved upstream eventually leading to fully separated flow conditions. In contrast to the stall mechanism at moderate Re , which

is governed by a separation bubble, stall initiated at the trailing edge results in a more gradual drop in lift, as can be observed by comparison of the numerical (fully turbulent boundary layer) and experimental (moderate Re : laminar/turbulent boundary layer) data in Figure 3.1.

It is suggested that the effects of separation bubbles and the importance of the laminar to turbulent transition on the hydrofoil's performance will be relatively less important in large scale laboratory and field tests. In reality, blade roughness and free stream turbulence, which is believed to be particularly high in marine currents, see Osalusi et al. (2009), may invoke transition to turbulence very close to the blade's leading edge. Hence, the accuracy of this type of CFD simulation should improve as one moves towards simulation of the full scale problem and environment.

3.1.1.2 Spatial convergence & turbulence model tests

Spatial convergence tests were conducted to identify an optimal mesh size with regard to computational costs and a satisfactory degree of grid convergence of the simulations.

During a turbine revolution the blades of a cross-flow turbine may experience large as well as rapid variations in angle of incidence, α . Under such conditions the form of the boundary layer cannot be assumed. Hence, the viscous affected inner wall region must be resolved properly in order to enable accurate simulation of the flow within the boundary layer. To this end the performance of two turbulence models with low Re capabilities, Spalart-Allmaras (S-A) and SST $k - \omega$, were investigated; for detailed information on both turbulence models see Section 2.2.2. For proper boundary layer resolution with these models it is desirable that the wall normal resolution is such that $y^+ = O(1)$ for the wall adjacent elements, see ANSYS Inc. (2009), where:

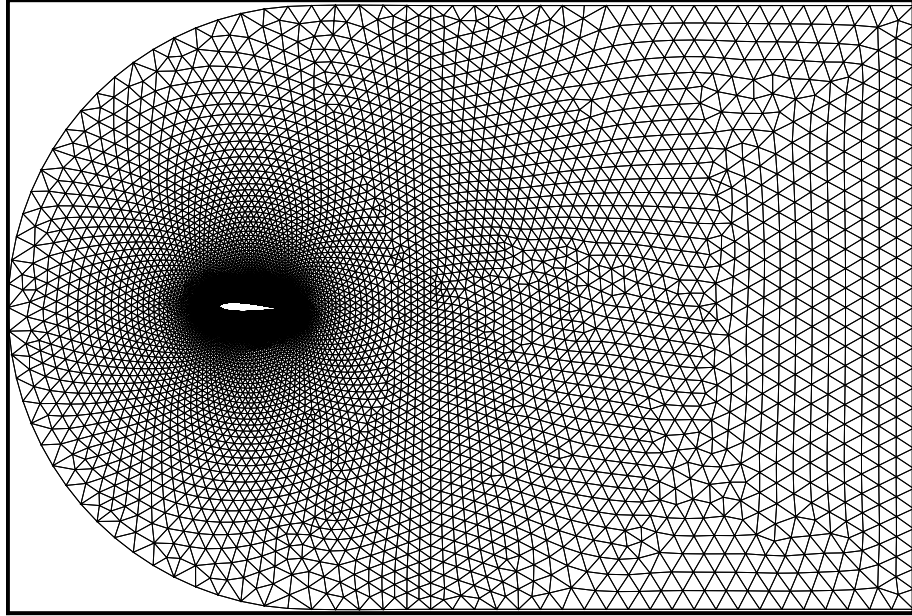
$$y^+ = \frac{y}{\nu} \sqrt{\frac{\tau_w}{\rho}} \quad \tau_w = \mu \left(\frac{\partial u}{\partial y} \right)_{y=0} \quad (3.1.3)$$

where τ_w , ρ , u and y are respectively the wall shear stress, fluid density, the flow velocity parallel to the wall and the wall normal distance to the centre of the first cell.

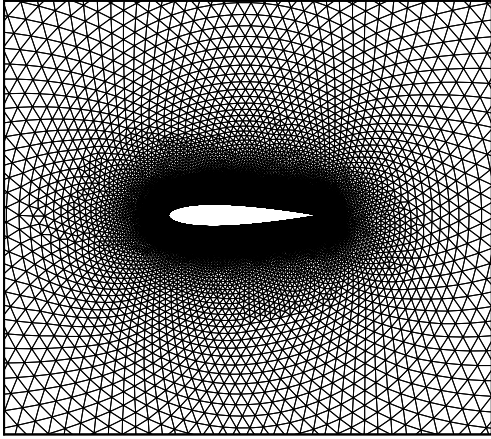
The computational domain for the static blade tests is made up of two sub-domains:

1. a far-field domain and
2. an inner circular domain around the blade.

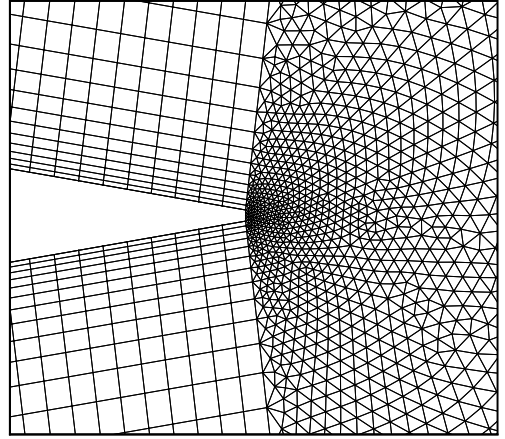
The full domain shown in Figure 3.2a extends 4 c upstream and 11 c downstream of the centre of the turbine and 5 c laterally to either side of the turbine, where $c = 1 \text{ m}$. The grid used was a hybrid mesh composed of a very fine structured grid for the near-wall region and unstructured grids for the remainder of the domain, as shown in Figure 3.2. While the far-field domain size was kept constant at 7,000 cells, the inner circular domain, shown in Figure 3.2b, was varied in size from 40,000 to 94,000 and 194,000, where the respective first grid spacing from the surface in the normal direction was $2.90 \times 10^{-4} c$, $1.45 \times 10^{-4} c$ and $7.25 \times 10^{-5} c$, resulting in $y^+ = 6.0$ (Mesh 1), 3.0 (Mesh 2), 1.5 (Mesh 3) at $\alpha = 2.5^\circ$; the growth factor employed for the wall normal distribution was 1.2. The streamwise mesh resolution was kept constant at 744 cells per blade side, where 644 cells are distributed uniformly over the front 90% of the blade and 100 cells over the rear 10% of the blade, resulting in the required increased mesh resolution at the trailing edge. The blades used for the simulations of the present study had a trailing edge thickness of 0.04% of the blade chord, see Figure 3.2c.



(a) Full domain



(b) Inner circular domain around blade



(c) Mesh around blade's trailing edge

Figure 3.2: Computational domain for static blade tests at $Re_c = 3.6 \times 10^5$

As to the boundary conditions the following conditions have been employed in the simulations:

1. no slip condition at the blades' surfaces, i.e. $U = 0 \text{ m/s}$ on the body surface,
2. unperturbed streamwise flow conditions across the upstream and cross-stream boundaries; streamwise free-stream flow velocity, $U_{\infty,x} = 1 \text{ m/s}$, cross-stream free-stream flow velocity, $U_{\infty,y} = 0 \text{ m/s}$, and
3. a constant pressure along the outflow of the downstream boundary, i.e. $P = 0 \text{ Pa}$;

further $\rho = 1.225 \text{ kg/m}^3$ and $\mu = 3.403 \times 10^{-6} \text{ kg/m}\cdot\text{s}$ resulting in $Re_c = 3.6 \times 10^5$. The turbulent intensity, Tu , was set to 2% at the inlet and 5% at the outlet; the turbulent viscosity ratio, ε , to 2 at the inlet and 5 at the outlet, where:

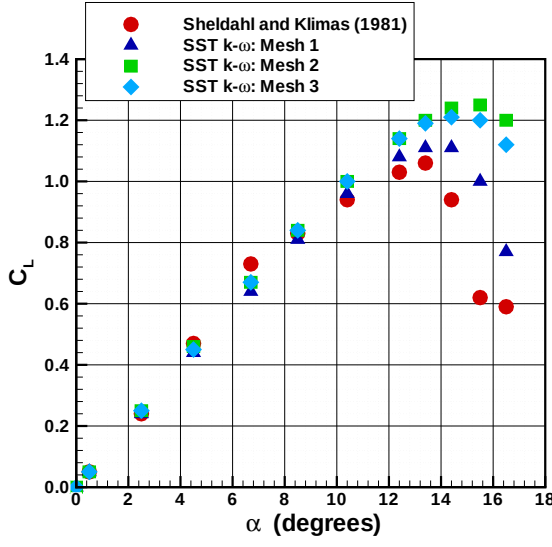
$$Tu = \frac{\left(\frac{2}{3}k\right)^{1/2}}{U_{ref}} \quad (3.1.4)$$

$$\varepsilon = \frac{\mu_t}{\mu} \quad (3.1.5)$$

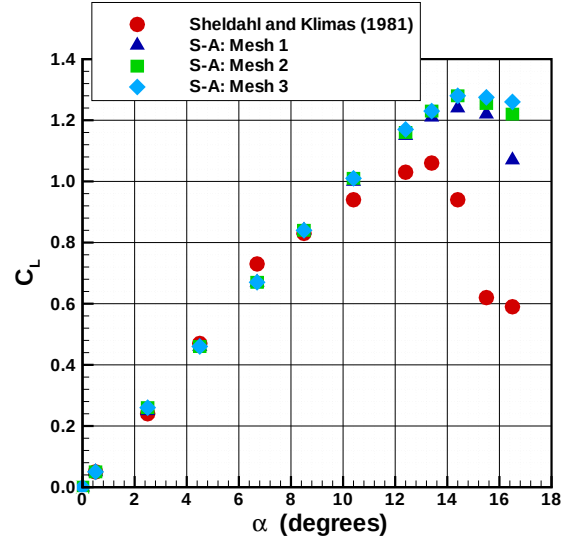
where k , U_{ref} , μ_t , μ are the turbulence kinetic energy, the mean velocity (Reynolds averaged), as discussed in Section 2.2.1, the turbulent viscosity and (molecular) dynamic viscosity, respectively.

Regarding the model discretisation, the PISO scheme was employed for pressure-velocity coupling, second-order upwind for the momentum and turbulence modelling equations, for more information see Chapter 2 and ANSYS Inc. (2009).

Figure 3.3 & 3.4 show the results of the computed lift and drag coefficients, C_L & C_D , for the three different meshes using both the Spalart-Allmaras and SST $k-\omega$ turbulence models as well as the results from the physical experiments conducted at the Sandia National Laboratories (SNL), USA, see Sheldahl and Klimas (1981).

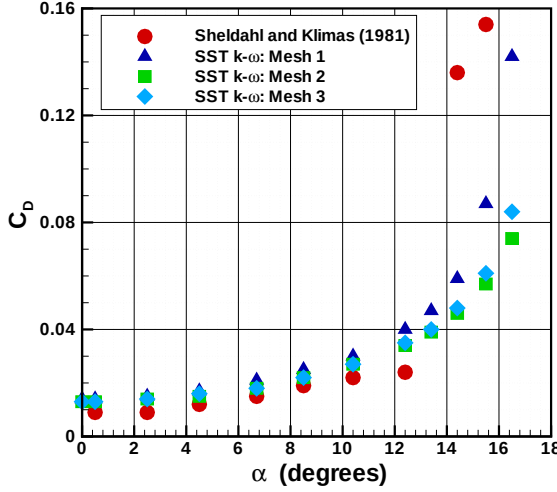


(a) Turbulence model: SST $k - \omega$

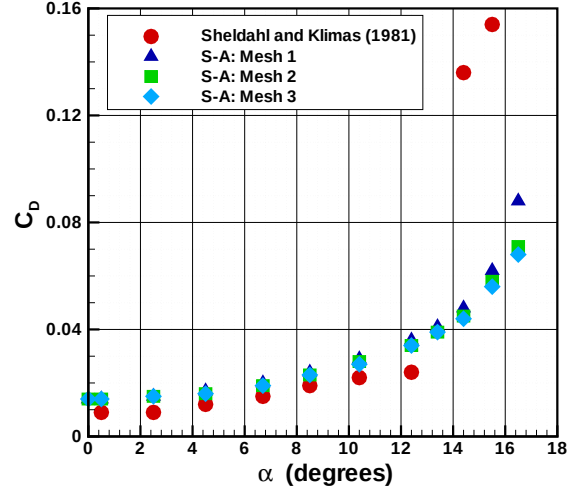


(b) Turbulence model: Spalart-Allmaras (S-A)

Figure 3.3: Grid convergence tests for static blade simulations at $Re_c = 3.6 \times 10^5$: lift coefficient



(a) Turbulence model: SST $k - \omega$



(b) Turbulence model: Spalart-Allmaras (S-A)

Figure 3.4: Grid convergence tests for static blade simulations at $Re_c = 3.6 \times 10^5$: drag coefficient

As to the grid convergence tests, Figure 3.3 & 3.4 show that the reduction in y from $2.90 \times 10^{-4} c$ (Mesh 1) to $1.45 \times 10^{-4} c$ (Mesh 2) has a significant effect on the lift and drag coefficients obtained from the simulations conducted with both turbulence models. For the attached flow region, the difference in C_L between Mesh 1 & Mesh 2 is up to 10% and the difference in C_D up to 15%.

A further reduction in y from $1.45 \times 10^{-4} c$ (Mesh 2) to $7.25 \times 10^{-5} c$ (Mesh 3) leads to a maximum change of 1% in C_L and 3% in C_D for $\alpha < 13^\circ$, which are esteemed acceptable. We consider the results to be sufficiently converged, when the change in C_L is smaller than about 1% and the change in C_D smaller than 5%. Accepting a larger error in C_D is justified by considering the driving mechanism of a cross-flow turbine, as given by Equation 3.1.6:

$$Q = BR(L \sin \alpha - D \cos \alpha) \quad (3.1.6)$$

where Q , B and R are the torque per unit length of blade, number of blades and the radius at the blade's tip.

It is acknowledged that in regions of high drag, such as flow separations, drag plays an important role in determining the instantaneous torque, but for the majority of a turbine revolution $L \sin \alpha$ is much larger than $D \cos \alpha$. Hence, the accuracy of the lift force is considered more important and an error in C_D of 5% is considered acceptable, if permitting a significant increase in the speed of the simulations. Due to the significant reduction of the computational costs of the simulations when using Mesh 2 instead of Mesh 3, it was hence decided to use Mesh 2 for all further simulations, where $Re_c \leq O(10^5)$, see Section 3.1.2 & 4.1; also the results from the present simulations shown in Figure 3.1 were computed using Mesh 2.

Regarding the differences between the simulations using the S-A and SST $k - \omega$ turbulence models, the latter are found to give slightly better force statistics below stall.

The numerically computed values of C_L at incidences below the static stall angle, α_{ss} , were very similar for both turbulence models, but the average error for C_D at $\alpha < \alpha_{ss}$ was 8% lower for the simulations using the SST $k - \omega$ than the S-A model.

The principal reason why the SST $k - \omega$ turbulence model was adopted for the full turbine simulations is due to the results for the post-stall region. The errors in C_L and C_D for $\alpha > \alpha_{ss}$ are significantly larger for the computations using the S-A model than those when using the SST $k - \omega$ model. This behaviour is expected, as the SST $k - \omega$ model is observed to be more adept at simulating grossly separated flows, see for example Tucker (2006).

At high angles of attack, $\alpha > 30^\circ$ say, the simulations using the SST $k - \omega$ model generated a periodic vortex wake, which was accompanied by periodic blade lift and drag forces, see Figure 3.5.

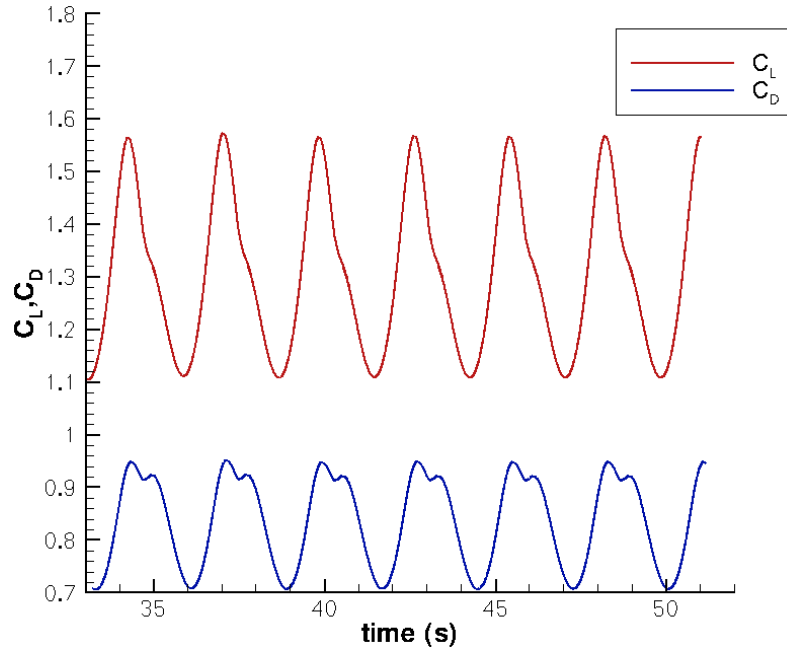


Figure 3.5: Periodic lift and drag histories for a static blade at $\alpha = 30.8^\circ$ using the SST $k - \omega$ turbulence model

In contrast, the simulations using the S-A model resulted in erratic blade force histories. Aerofoil flows have been shown to exhibit chaotic flow patterns, for example see Barton

and Pulliam (1986) or Pulliam and Vastano (1990), and the SST $k - \omega$ model may in fact be over-dissipative. Since the force histories from the experimental results are not available, it is difficult to deduce, whether the simulations using the S-A model are identifying real fluid mechanics.

However, as discussed above, it is apparent that the simulations using the SST $k - \omega$ model render lower errors in C_L and C_D than the simulations using the S-A model relative to the experimental results. Hence, it was decided to use the SST $k - \omega$ turbulence model for all further simulations and Mesh 2, with $y^+ = 3$ at $\alpha = 2.5^\circ$, for simulations, where $Re_c \leq O(10^5)$.

3.1.2 Rotating turbine tests

In addition to the static blade tests, it is important to validate any dynamic effects arising from the continuous changes in blade loading typical of cross-flow turbines. Due to the lack of available data for oscillating blade tests at $Re_c = O(10^5)$ and lack of data for the loading experienced by a single blade of a straight-bladed cross-flow turbine throughout a cycle at a moderate Re , it was necessary to compare results from the present simulations to data from physical experiments carried out at $Re_c = O(10^4)$.

Two different turbine configurations have been simulated:

1. NACA 0015 one-bladed turbine with a turbine solidity, $\sigma = 0.040$, from Sandia National Laboratories (SNL) tests, see Strickland et al. (1981), operating at a tip speed ratio, $\lambda = 5.1$, and $Re_c = 6.7 \times 10^4$;
2. NACA 0018 two-bladed turbine with $\sigma = 0.064$ from Sherbrooke University tests, see Vittecoq and Laneville (1982), operating at $\lambda = 5.0$ and $Re_c = 3.8 \times 10^4$; where

$$\sigma = \frac{cB}{\pi 2R} \tag{3.1.7}$$

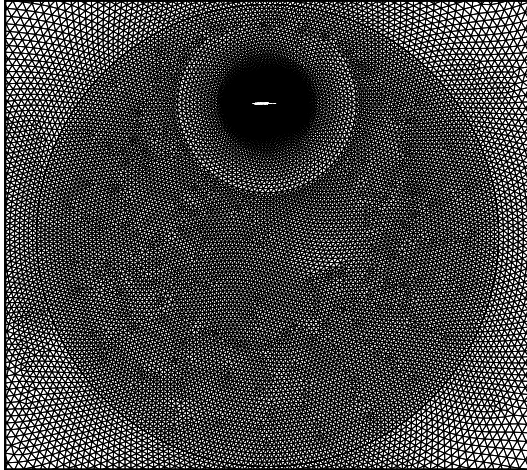
$$\lambda = \frac{\omega R}{U_\infty} \quad (3.1.8)$$

where ω is the turbine angular velocity.

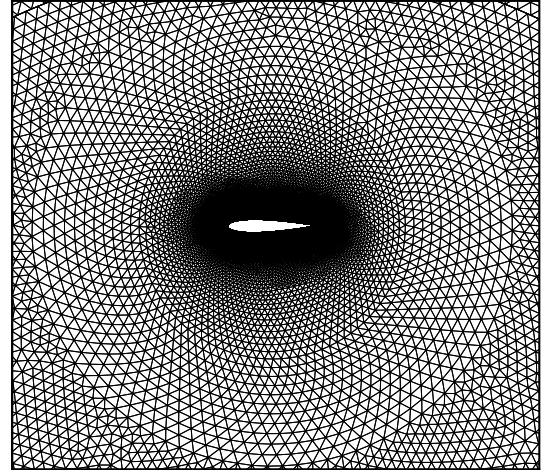
The set-up of the numerical model was based on the settings employed for the static blade tests discussed in Section 3.1.1. The model discretisation and boundary conditions were the same; $U_\infty = 1 \text{ m/s}$ at inflow, $P = 0 \text{ Pa}$ at outflow. μ was altered to achieve the required Reynolds number; $\mu = 9.142 \times 10^{-5} \text{ kg/m} \cdot \text{s}$ for the one-bladed turbine resulting in $Re_c = 6.7 \times 10^4$ and $\mu = 1.644 \times 10^{-4} \text{ kg/m} \cdot \text{s}$ for the two-bladed turbine resulting in $Re_c = 3.8 \times 10^4$.

The computational domain, shown in Figure 3.6, is a two-dimensional slice orthogonal to the turbine's axis of rotation; it is made up of three sub-domains:

1. a far-field domain,
2. a turbine domain consisting of a circular rotating mesh and
3. discrete circular domains around each blade, where $c = 1 \text{ m}$.



(a) Turbine domain - One-bladed turbine



(b) Inner circular domain

Figure 3.6: Computational domain for rotating turbine tests

For the NACA 0015 one-bladed turbine, Mesh 2 from the static blade tests was used for the discrete circular domain around the blade, and for the NACA 0018 two-bladed turbine, a similar mesh, where, likewise, $y = 1.45 \times 10^{-4} c$, was employed.

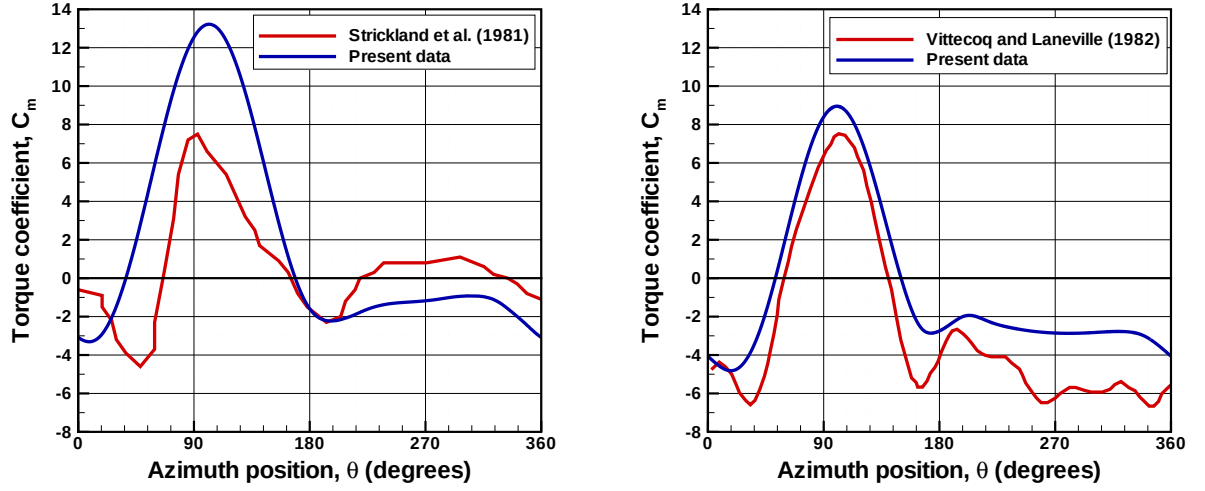
Grid convergence tests were also performed for the NACA 0018 mesh following the same approach as for the static blade tests for the NACA 0015 blade outlined in Section 3.1.1. For the attached flow region, a reduction in y from $1.45 \times 10^{-4} c$ to $7.25 \times 10^{-5} c$ resulted in a maximum change of 1% in C_L and 2% in C_D , which was considered satisfactory.

The total number of elements for the one-bladed turbine mesh is 140,000 and 245,000 for the two-bladed turbine. In both cases, the domain extends $8 R$ upstream and $22 R$ downstream of the centre of the turbine and $8 R$ laterally to either side of the turbine centre. Because the computational domain is two-dimensional, the turbine is implicitly assumed to be infinitely long. To simulate the rotation of the rotor, the circular turbine mesh with embedded blades is prescribed to move relative to the outer inertially fixed domain.

Figure 3.7 shows plots of the torque coefficient, C_m , of an individual blade of each of the two turbines against the azimuth position, θ , of the blade, where:

$$C_m = \frac{Q}{\frac{1}{2}\rho U_\infty^2 c^2} \quad (3.1.9)$$

is the non-dimensional representation of the blade torque per unit span, Q .



(a) Sandia National Laboratories tests: $\sigma = 0.040$, $B = 1$, $\lambda = 5.1$, $Re_c = 6.7 \times 10^4$

(b) Sherbrooke University tests: $\sigma = 0.064$, $B = 2$, $\lambda = 5.0$, $Re_c = 3.8 \times 10^4$

Figure 3.7: Comparison of numerically and experimentally obtained blade torque coefficient traces

The azimuth angle, $\theta = 0^\circ$, corresponds to a blade's top position vertically above the centre of the turbine, see Figure 1.12b, and thus an angle of incidence, α , of 0° assuming no flow perturbation by the blade. When studying the results of the physical experiments and of the present numerical investigation, it is seen that all four torque traces in Figure 3.7 exhibit the expected shape. As the blade reaches $\theta \approx 90^\circ$ position and thus maximum angle of incidence, the first peak in the torque traces are observed; at around $\theta = 180^\circ$, C_m reaches a minimum. Through the downstream passage, it is evident that the blades contribute little to no positive torque, because they operate in perturbed flow conditions - the wakes developed on upstream blade passages. The performance of the blades on the downstream passes depends on the degree of flow impedance and perturbation, which is dependent on turbine solidity, σ , and tip speed ratio, λ . Moreover, this may explain the inferior performance for $180^\circ < \theta < 360^\circ$ of an individual blade of the turbine configuration tested at Sherbrooke University, because the turbine has more blades and a higher turbine solidity than the machine tested at

the Sandia National Laboratories.

When comparing the numerical and experimental traces in Figure 3.7a & 3.7b, the key difference between the numerically and experimentally obtained torque data is the over-prediction of C_m by the numerical model on the upstream passage of the turbine. This may be attributed to two factors:

1. As discussed in Section 3.1.1, the CFD model simulates a different stalling mechanism leading to a higher static stall angle and hence higher maximum lift and thus higher torque.
2. In the present work the blades are treated as infinitely long and the flow computed as two-dimensional. No account is taken of the supporting struts present in the physical experiments. Strut drag will reduce turbine torque directly by providing a negative torque and indirectly by increasing turbine thrust resulting in an increase in streamwise flow impedance and thus a reduction of the angle of attack experienced by the blades, and hence lower blade lift and torque; from Figure 1.12b, it is apparent that a reduction in the streamwise flow velocity, U_x , results in a reduction of α .

3.2 Reynolds number = $O(10^6)$

Furthermore, we are interested in real turbine flows and conducted turbine simulations at $Re_c = O(10^6)$, presented in Section 4.2 and Chapter 5. Hence, the static blade tests were repeated at $Re_c = 6.8 \times 10^5$, for which experimental data is also available from Sheldahl and Klimas (1981), and, in addition, experimental data is available from Piziali (1994) for oscillating blade tests at $Re_c = 2 \times 10^6$, which were repeated numerically in order to validate the unsteady blade loadings typical of a cross-flow turbine.

3.2.1 Static blade tests

As above, we first present the results from the converged numerical solution to discuss the flow physics for the static blade tests conducted at $Re_c = 6.8 \times 10^5$, see Section 3.2.1.1; subsequently, in Section 3.2, the sensitivity of the present dynamic blade simulations to mesh refinement is discussed.

3.2.1.1 Comparison of numerically and experimentally obtained lift and drag data

The numerical settings of the static blade tests conducted at $Re_c = 6.8 \times 10^5$ were identical to the ones described in Section 3.1.1. As in the previous simulations, the boundary conditions employed were $U_\infty = 1 \text{ m/s}$ at inflow, $P = 0 \text{ Pa}$ at outflow. The only parameters changed were the mesh and the fluid's kinematic viscosity, $\nu = 1.801 \times 10^{-6} \text{ m}^2/\text{s}$ resulting in $Re_c = 6.8 \times 10^5$, for which lift and drag data is available from the same SNL test series, as used in Section 3.1.1, see Sheldahl and Klimas (1981); the higher the Re , the smaller the smallest turbulence length scales and hence the need for a finer mesh. The grid employed for the simulations at $Re_c = 6.8 \times 10^5$ is discussed in more detail in Section 3.2, but one of the key differences to the grid used for the

static blade tests at $Re_c = 3.8 \times 10^4$ is the reduction in the first grid spacing from the surface in the normal direction from $1.45 \times 10^{-4} c$ to $5 \times 10^{-5} c$.

Figure 3.8 shows the lift and drag data from the present numerical investigation for a NACA 0015 blade operating at $Re_c = 3.6 \times 10^5$ & 6.8×10^5 as well as the corresponding experimental data from SNL tests found in Sheldahl and Klimas (1981).

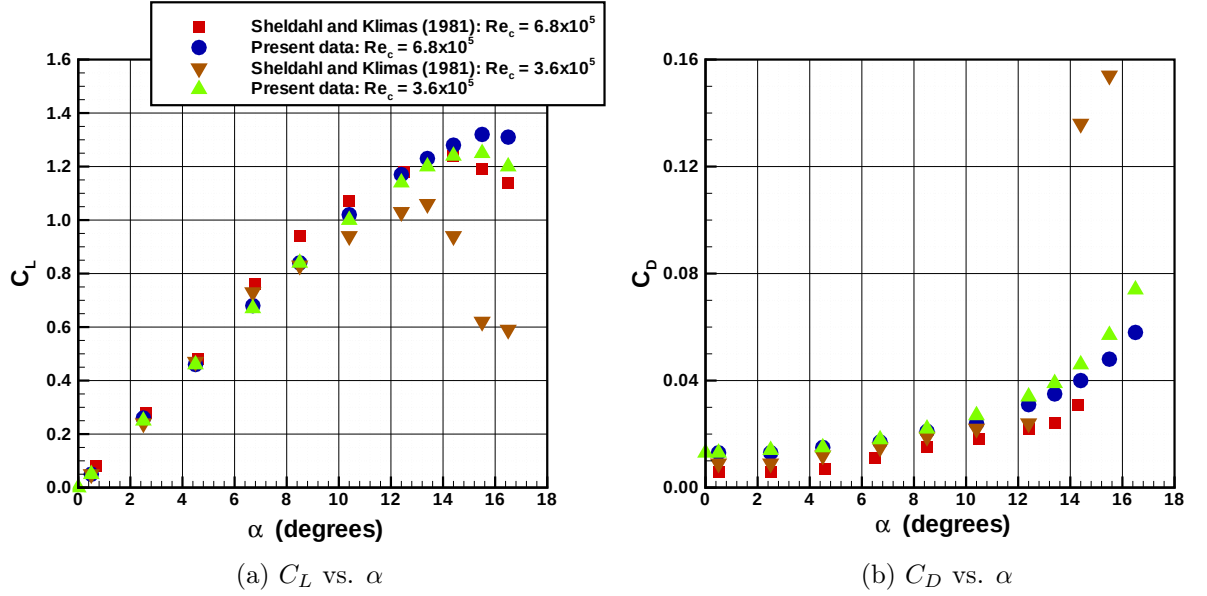


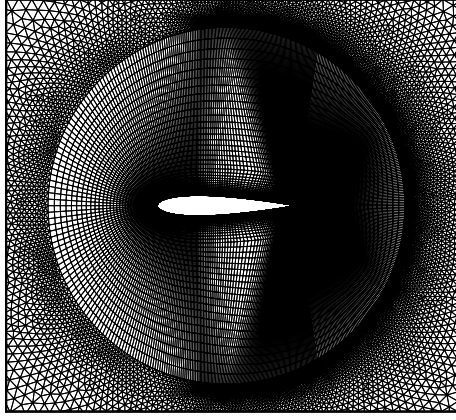
Figure 3.8: NACA 0015 blade at $Re_c = 3.6 \times 10^5$ & 6.8×10^5 : lift and drag coefficients

When comparing the lift data for $Re_c = 6.8 \times 10^5$ from Sheldahl and Klimas (1981) and the numerical tests, it may be deduced that the differences discussed in Section 3.1.1 still persist, but are not as distinct due to the increase in Re . Whilst the CFD model over-predicts the peak in max C_L by about 2 degrees and max C_L by around 20% at $Re_c = 3.6 \times 10^5$, the peak in max C_L is over-predicted by 1 degree and max C_L by around 6% at $Re_c = 6.8 \times 10^5$.

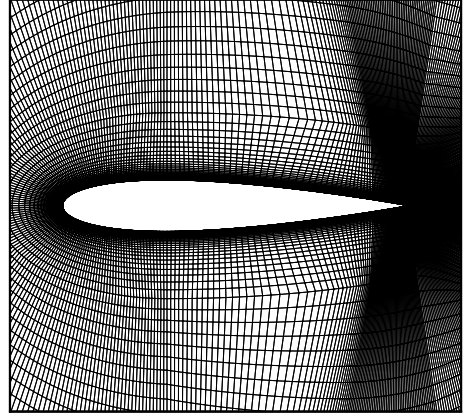
Similarly, the differences in C_D from the numerical and experimental tests for $13^\circ < \alpha < 15^\circ$ are smaller for the higher Re simulations, as the sudden increase in drag associated with flow separation occurs at higher incidence; at $\alpha = 14.4^\circ$, experimentally derived $C_D = 0.136$ at $Re_c = 3.6 \times 10^5$, whilst $C_D = 0.031$ at $Re_c = 6.8 \times 10^5$. This

higher degree of agreement between the numerical and experimental tests observed at $Re_c = 6.8 \times 10^5$ is to be expected, because the size and effect of the laminar separation bubble, discussed in Section 3.1.1, decreases with increasing Re ; recalling that the simulations use turbulent flow throughout.

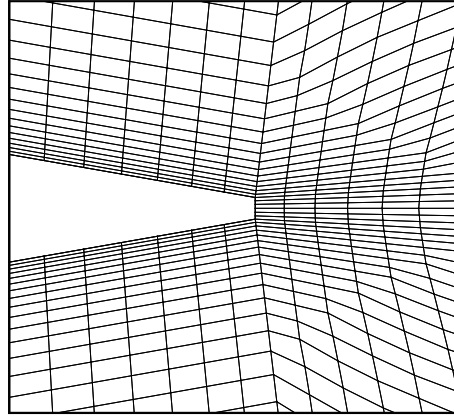
3.2.1.2 Spatial convergence tests



(a) Circular domain around blade



(b) Blade domain - close-up



(c) Mesh around blade's trailing edge

Figure 3.9: Computational domain for static blade tests at $Re_c = 6.8 \times 10^5$

The grid used for these simulations was a hybrid mesh comprised of a very fine structured quadrilateral element grid for the near-wall region, a fine structured grid for the circular region within $1.5\ c$ of the hydrofoil and an unstructured triangular grid for the remainder of the domain, see Figure 3.9.

Due to the increase in the total number of elements and the decrease in size of the individual cells, in particular in close proximity to the blade's surface, associated with the higher Re simulations, it was paramount to find the optimal mesh size with regard to computational costs and a satisfactory degree of convergence of the simulations. It was hence decided to conduct a spatial convergence study, firstly, for the chordwise resolution and, secondly, for the direction normal to the blade's surface.

Chordwise resolution

The first six meshes tested all had a first grid spacing from the surface in the normal direction of $1 \times 10^{-5}\ c$, which results in a maximum $y^+ = 0.35$ at $\alpha = 2.5^\circ$, but had a varying chordwise resolution. In fact, the blade was split into three sections in the chordwise direction. As the leading and trailing edges require a higher mesh resolution, the three sections extend from (I) $0\ c - 0.3\ c$, (II) $0.3\ c - 0.9\ c$ and (III) $0.9\ c - 1\ c$, see Figure 3.10.

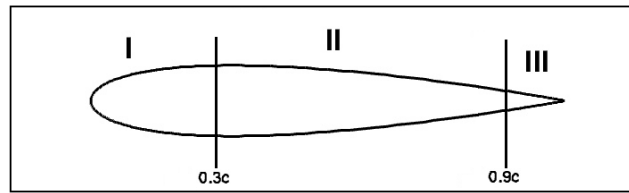


Figure 3.10: Blade resolution regions

The extensive investigation, whose results are summarised in Table 3.2, aimed at optimising the chordwise mesh resolution for the individual sections.

Section	I	II	III
Cells	200	320	100
	α	C_L	C_D
	0.5°	0.05	0.013
	2.5°	0.25	0.013
	4.5°	0.46	0.015
	6.7°	0.67	0.017
	8.5°	0.84	0.021
	10.4°	1.01	0.025
	12.4°	1.17	0.031

(a) Mesh I: 170,000 cells

I	II	III
100	160	60
α	Δ in C_L	Δ in C_D
0.5°	0.0%	0.1%
2.5°	0.1%	0.1%
4.5°	0.1%	0.2%
6.7°	0.2%	0.3%
8.5°	0.2%	0.5%
10.4°	0.3%	1.1%
12.4°	0.4%	1.6%

(b) Mesh II: 110,000 cells

I	II	III
100	160	30
α	Δ in C_L	Δ in C_D
0.5°	3.2%	0.1%
2.5°	2.9%	0.2%
4.5°	2.3%	0.3%
6.7°	1.9%	0.4%
8.5°	1.9%	0.8%
10.4°	1.8%	1.5%
12.4°	1.7%	1.9%

(c) Mesh III: 80,000 cells

I	II	III
50	160	60
α	Δ in C_L	Δ in C_D
0.5°	1.9%	3.8%
2.5°	2.4%	4.9%
4.5°	2.4%	6.5%
6.7°	2.5%	8.2%
8.5°	2.9%	8.9%
10.4°	3.2%	10.9%
12.4°	3.5%	12.2%

(d) Mesh IV: 100,000 cells

I	II	III
100	80	60
α	Δ in C_L	Δ in C_D
0.5°	0.1%	0.2%
2.5°	0.2%	0.3%
4.5°	0.3%	0.5%
6.7°	0.3%	0.9%
8.5°	0.5%	1.4%
10.4°	0.6%	2.1%
12.4°	0.8%	2.5%

(e) Mesh V: 90,000 cells

I	II	III
70	40	50
α	Δ in C_L	Δ in C_D
0.5°	0.3%	1.1%
2.5°	0.4%	1.5%
4.5°	0.4%	1.6%
6.7°	0.5%	1.7%
8.5°	0.7%	2.1%
10.4°	0.8%	2.6%
12.4°	1.0%	3.4%

(f) Mesh VI: 75,000 cells

Table 3.2: Changes in lift and drag computations using meshes with different chordwise discretisations and $y = 1 \times 10^{-5} c$

Table 3.2a shows the computed lift and drag data for the finest mesh (Mesh I) tested, for which section I comprises 200 cells, section II 320 cells and section III 100 cells in the chordwise direction per blade side, resulting in 170,000 elements for the circular mesh around the blade, shown in Figure 3.9a. Table 3.2b - 3.2f also show the number of cells per blade section as well as the difference in the computed lift and drag coefficients to the results obtained for Mesh I.

As for the static blade tests in Section 3.1.1, the present author considers the results sufficiently converged, when the change in C_L is smaller than about 1% and the change in C_D smaller than 5%. The spatial convergence of Mesh I is confirmed by halving the

edge length, see results for Mesh II, and hence it is valid to compute ΔC_L and ΔC_D relative to Mesh I. The meshes III, IV, and V are systematic regional refinements on Mesh II to identify where resolution is required.

The comparison shows that a reduction to 50 cells in section I leads to errors larger than 3% in C_L and larger than 12% in C_D for $\alpha < 13^\circ$, see Table 3.2d; a reduction to 30 cells in section III results in errors of up to 3.2% in C_L for $\alpha < 13^\circ$, see Table 3.2c, which the present author all esteemed too large.

The results from these investigations led to mesh VI, which fulfils the grid convergence criteria and imposes the lowest computational costs, see Table 3.2f; the maximum error in C_L is 1.0% and in C_D smaller than 4% for $\alpha < 13^\circ$. It was hence decided to conduct all further simulations at $Re_c = O(10^6)$ with a chordwise resolution, as described in Table 3.2f.

Normal resolution

The next tests were used to identify an appropriate mesh resolution in the direction normal to the blade's surface. Table 3.3 shows the differences in the computed lift and drag coefficients for three different meshes with varying y^+ values in comparison to the results obtained with Mesh I, for which $\max y^+ = 0.35$ at $\alpha = 2.5^\circ$. All three meshes had the same chordwise resolution as Mesh VI, but the first grid spacing from the surface in the normal direction was increased to $5 \times 10^{-5} c$, $1 \times 10^{-4} c$ and $5 \times 10^{-4} c$ respectively, resulting in $\max y^+$ values of 1.6 (Mesh VI_a), 3.2 (Mesh VI_b) and 15.0 (Mesh VI_c) at $\alpha = 2.5^\circ$; the growth factor employed for the wall normal distribution was $\max 1.2$.

α	Δ in C_L	Δ in C_D
0.5°	0.7%	5.6%
2.5°	0.8%	5.5%
4.5°	0.9%	5.4%
6.7°	0.9%	5.3%
8.5°	1.0%	4.8%
10.4°	1.1%	4.4%
12.4°	1.2%	3.9%

α	Δ in C_L	Δ in C_D
0.5°	1.3%	8.0%
2.5°	1.2%	7.9%
4.5°	1.1%	7.2%
6.7°	1.1%	6.1%
8.5°	0.7%	4.1%
10.4°	0.7%	3.5%
12.4°	0.6%	2.9%

α	Δ in C_L	Δ in C_D
2.5°	2.6%	24.0%
6.7°	2.9%	22.3%
10.4°	3.7%	18.4%

(a) Mesh VI_a: $y = 5 \times 10^{-5} c$: 50,000 cells (b) Mesh VI_b: $y = 1 \times 10^{-4} c$: 47,000 cells (c) Mesh VI_c: $y = 5 \times 10^{-4} c$: 41,000 cells

Table 3.3: Changes in lift and drag computations using meshes with different wall normal discretisations

Table 3.3c shows unsatisfactorily large errors for the attached flow region in both C_L and C_D at up to 4% and of over 20% respectively for Mesh VI_c. For Mesh VI_b the maximum difference in C_L is 1.3% and 8.0% in C_D for $\alpha < 13^\circ$, which is also considered too large; for Mesh VI_a the maximum difference in C_L is 1.2% and 5.6% in C_D .

When comparing the computation time of the static blade simulations using the different meshes, it is apparent that the employment of Mesh VI_a allows for a significant reduction in the computational costs in comparison to Mesh VI. Moreover, the maximum error of up to 5.6% in C_D observed for Mesh VI_a is relatively small when comparing the differences between the drag forces predicted by the present CFD model and recorded for the physical experiments, see Figure 3.8b. Hence, based on the static blade tests, Mesh VI_a with $\max y^+ = 1.6$ at $\alpha = 2.5^\circ$ is considered most suitable for further simulations at $Re_c = O(10^6)$, although the max error in C_D for $\alpha < 13^\circ$ is slightly larger than initially considered acceptable.

Moreover, it is interesting to note that Mesh VI_b exhibits a better performance than Mesh VI_a at higher angles of attack. This may be due to a better resolution of the boundary layer height by changing the first grid spacing from the surface in the normal direction. Increasing y in the first cell will increase the boundary layer resolution thickness, which may improve the capture of boundary layers at high α .

3.2.2 Dynamic blade tests

For the final stage of the validation two sets of oscillating blade tests were conducted at $Re_c = 2 \times 10^6$. Dynamically simulated lift and drag data for a 2D NACA 0015 aerofoil oscillating by $\pm 4^\circ$ around a fixed pitch angle of 4° and 13° was compared to experimental wind-tunnel data from Piziali (1994).

The set-up of these simulations was based on the static blade tests outlined in the previous section. In order to simulate the dynamic motion of the blade, a sinusoidal motion is prescribed to the circular domain with the blade imbedded, shown in Figure 3.9a:

$$\alpha(t) = \alpha_I + \alpha_0 \sin(\omega_f \times t) \quad (3.2.1)$$

$$\Omega(t) = \alpha_0 \omega_f \cos(\omega_f \times t) \quad (3.2.2)$$

where α , α_I , α_0 , ω_f , t and Ω are the pitch angle, initial incidence, pitch amplitude, angular frequency, time and angular velocity respectively.

The reduced frequency, k , for both tests, where $\alpha = 4^\circ \pm 4^\circ$ and $\alpha = 13^\circ \pm 4^\circ$, was 0.10, where:

$$k = \frac{\omega_f c}{2U_\infty} \quad (3.2.3)$$

To match the conditions of the physical experiments, where $k = 0.10$, and given that $c = 1 \text{ m}$ and $U_\infty = 1 \text{ m/s}$ for the numerical tests, $\omega_f = 0.2 \text{ rad/s}$ and $\alpha_0 = 0.0698 \text{ rad}$.

For both oscillatory blade tests, blade incidence was first increased from the fixed pitch angle to the maximum angle of attack before being reduced to the minimum angle of attack and then back to the fixed pitch angle; $\alpha = \{4^\circ \rightarrow 8^\circ \rightarrow 0^\circ \rightarrow 4^\circ\}$ and $\alpha = \{13^\circ \rightarrow 17^\circ \rightarrow 9^\circ \rightarrow 13^\circ\}$; the blades were oscillated about the quarter-chord. As

to the simulation length, up to 3 cycles were required to reach periodicity.

Due to the increase in Re_c to 2×10^6 from that used in the static blade tests ($Re_c = 6.8 \times 10^5$) as well as potential changes in the flow physics associated with a dynamic flow problem, some of the grid convergence tests were repeated for the dynamic blade simulations; these are discussed in Section 3.2.2.2.

3.2.2.1 Numerically and experimentally obtained oscillatory aerofoil forces

Figure 3.11 shows a comparison of the lift and drag data for both oscillating tests from physical experiments carried out by Piziali (1994), the present numerical investigation and another numerical investigation conducted by Gretton et al. (2009). The present CFD simulations were computed using Mesh VI_a and time step, $\Delta t = 1 \times 10^{-3} \text{ sec}$. The numerical investigation of Gretton et al. (2009) was conducted using a similar approach to the present study using an alternative commercial CFD code, ANSYS CFX, see ANSYS Inc. (2006), on a structured mesh.

For both tests and for both the numerically as well as experimentally determined blade forces, see Figure 3.11, the lift and drag plots exhibit hysteresis loops. For the $\alpha = 13^\circ \pm 4^\circ$ test, the lift curve is traced out in a clockwise direction, whilst the drag curve in an anti-clockwise direction, as is expected for blades operating passed the static stall angle, which results in a significant loss in lift and increase in drag and hence the observed loop orientations.

However, for the $\alpha = 4^\circ \pm 4^\circ$ tests stalling conditions are not reached and the hysteresis loop is a result of shed vorticity and it's effect on the incident flow-field. As the blade is dynamically pitched, vorticity is shed as a result of changes in the blade's bound vorticity, which adjusts the near-field causing the observed blade force hystereses; an anti-clockwise orientation for the lift and clockwise for the drag.

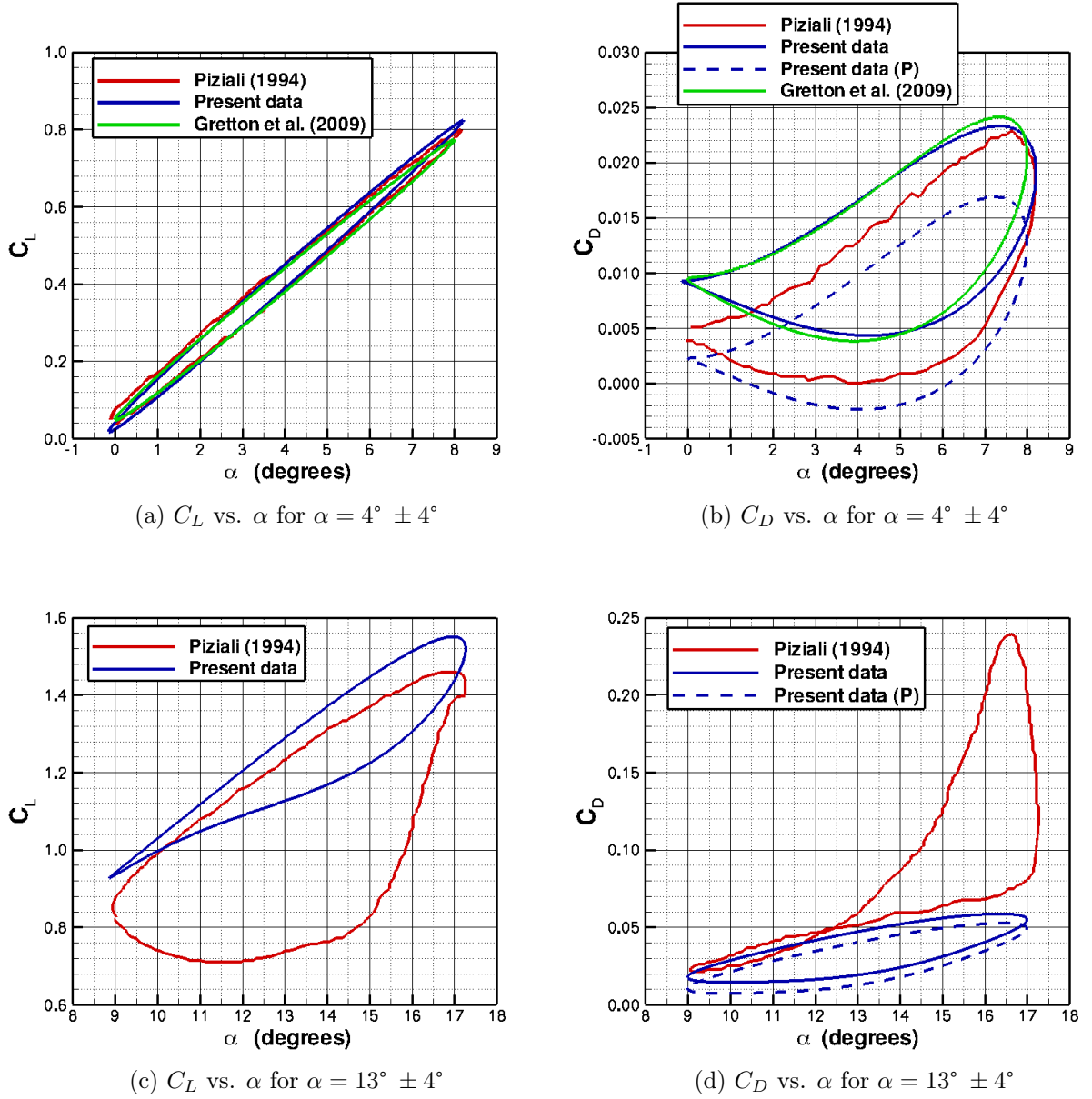


Figure 3.11: NACA 0015 blade at $Re_c = 2 \times 10^6$: oscillating blade tests : lift and drag coefficients

As incidence is increased, blade bound circulation increases, which results in shedding of oppositely signed vorticity from the trailing edge. The circulation of the near wake generates a downwards velocity at the blade resulting in a reduction of the effective angle of attack. This downwash has a similar effect to that observed for lift-induced drag; tilting the lift vector rearwards and reducing its magnitude, leading to a reduction

in lift and increase in drag. For the pitch down motion the reverse is the case, thus resulting in an anti-clockwise loop for lift and clockwise for drag.

When comparing the force predictions from the present investigation to the results from the wind-tunnel tests, the observations are principally similar to those made for the static blade tests in Section 3.1.1 & 3.2.1.

The lift coefficients for the $\alpha = 4^\circ \pm 4^\circ$ tests, thus the attached flow region, are well predicted. The widths of the hysteresis loops for the present and experimental data are similar, whilst the gradient of the loop of the present data is marginally higher, leading to an error of around 3% in C_L at $\alpha = 8^\circ$. The simulations carried out by Grettton et al. (2009) render similar results for C_L , but the gradient of the loop is marginally lower than that of the experimental data.

The drag coefficients for the $\alpha = 4^\circ \pm 4^\circ$ tests predicted by the present investigation and the study of Grettton et al. (2009) are similar, too, as shown in Figure 3.11b. Because the experimental drag coefficient was obtained from pressure tap measurements, the skin friction component of the computationally determined drag force should be neglected in order to make proper comparison with the experiments. The (P) in Figure 3.11b & 3.11d indicates pressure drag only.

In comparison to the experimental results, the widths of the numerical hysteresis loops agree well, but the present CFD model under-predicts the pressure drag, whereas it over-predicts the total drag for the static blade tests. The difference between total and pressure drag, the skin friction drag, may, to some extent, elucidate the contrary observations made, when comparing the computationally determined drag forces to the experimentally derived results for the static and dynamic blade tests.

As discussed in Section 3.1.1, the skin friction drag is over-estimated by the numerical simulation due to stipulating the boundary layer to be fully turbulent, thus leading to a significant over-prediction of the total drag for the attached flow region. However, the pressure drag may have been under-predicted by the CFD model for the static tests, too,

as it is the case for the dynamic blade tests. As only the total drag data is available for the static blade tests conducted by Sheldahl and Klimas (1981), this cannot be further investigated. Thus, it may not be concluded that the over-prediction of the total drag by the numerical model observed for the static blade tests is a contradictory result to the under-prediction of the pressure drag by the numerical model for the dynamic blade tests.

Moreover, it is observed that $C_D(P)$ computed by the present numerical model goes negative on the downward path, which may be explained as follows. On the downward path C_L decreases and hence positive vorticity is shed; this results in an upwash, so that lift vector forward is tilted forward. This can also be thought of as induced drag, which overcomes the real drag. It is the opposite of the downwash/induced drag argument presented above to explain the anti-clockwise orientation of the lift hysteresis loop for the $\alpha = 4^\circ \pm 4^\circ$ oscillatory blade tests.

With regard to the $\alpha = 13^\circ \pm 4^\circ$ dynamic blade tests, significant differences arise between the experimentally and computationally obtained forces. As for the static blade tests, the numerically derived lift coefficient is over-estimated, as α is increased from its initial position at 13° due to stipulating the boundary layer to be fully turbulent, which leads to a more benign (trailing edge) dynamic stalling mechanism than appears to have occurred in the physical experiments. This is consistent with the significantly less distinct hysteresis loops observed for the computationally obtained lift and drag coefficients.

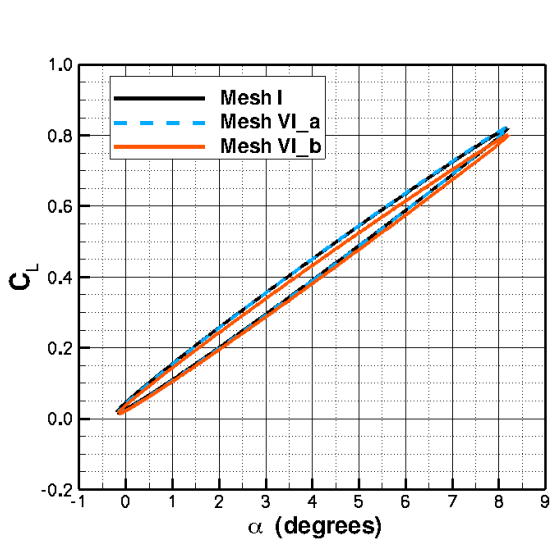
Also, C_D is under-estimated by the numerical model; this is due to the adverse effects of the separated flow region on drag being less significant for the computational model than for the real flow due to the differing stall behaviour.

3.2.2.2 Spatial convergence tests

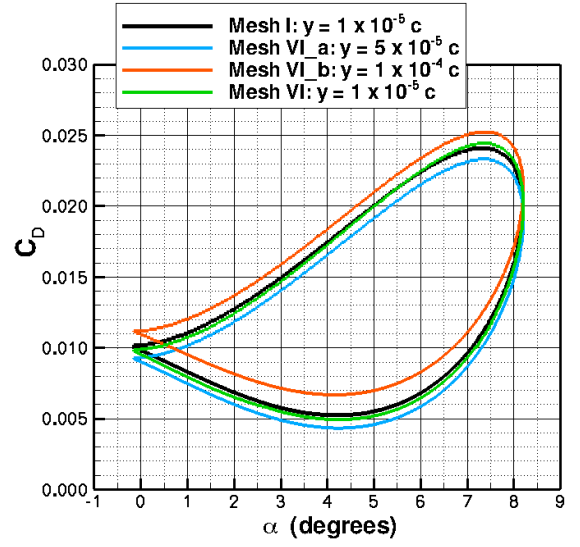
Figure 3.12a shows the computed lift coefficients, where $\alpha = 4^\circ \pm 4^\circ$, for the simulations using the mesh with the finest chordwise resolution and lowest y^+ tested, Mesh I, see Table 3.2a, and two with lower chordwise and wall normal resolutions; Meshes VI_a and VI_b, Table 3.3a & 3.3b. It is apparent that both the reduced chordwise resolution as well as the higher y of Mesh VI_a relative to Mesh I have little effect on the computed lift coefficients. However, a further increase in y to $1 \times 10^{-4} c$, as for Mesh VI_b, results in a lower gradient of the lift cycle.

When studying the corresponding drag plots in Figure 3.12b, the difference between the drag traces obtained using Mesh I and Mesh VI_b is more distinct than for the lift traces; Mesh VI_b over-predicts C_D by 20% to 30% for most of the cycle. The difference is reduced to less than 10% for the majority of the cycle when comparing the drag data for Mesh I and Mesh VI_a.

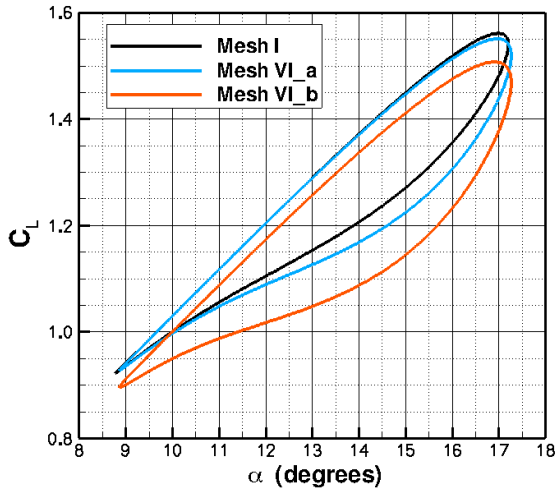
In addition, the drag coefficients computed using Mesh VI (lower chordwise resolution but same y as Mesh I) is plotted in Figure 3.12b and by comparison of the different force traces, it is apparent that the wall normal resolution is principally responsible for the differences in the drag coefficients simulated. However, given the significant increase in computation time when using Mesh VI relative to when using Mesh VI_a, the additional accuracy for C_D for the $\alpha = 4^\circ \pm 4^\circ$ tests due to employing Mesh VI is not considered sufficiently great by the present author.



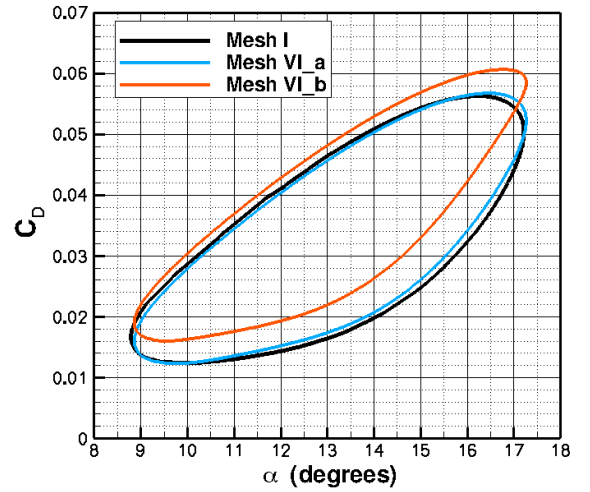
(a) C_L vs. α for $\alpha = 4^\circ \pm 4^\circ$



(b) C_D vs. α for $\alpha = 4^\circ \pm 4^\circ$



(c) C_L vs. α for $\alpha = 13^\circ \pm 4^\circ$



(d) C_D vs. α for $\alpha = 13^\circ \pm 4^\circ$

Figure 3.12: Grid convergence tests for oscillating blade simulations at $Re_c = 2 \times 10^6$

Similarly, Figure 3.12c & 3.12d show the lift and drag data for the oscillating blade tests, where $\alpha = 13^\circ \pm 4^\circ$. Both the lift and drag coefficients are unsatisfactorily predicted by the simulations using Mesh VI_b; the maximum errors of over 8% in lift and over 25% in drag in comparison to the results for Mesh I are considered too large. In contrast, the lift and drag traces for Mesh VI_a show a significantly higher incidence of agreement with the corresponding data for Mesh I. The drag coefficient is well pre-

dicted by the computations with Mesh VI_a for the entire cycle, whilst the lift force is well predicted for a large part of the cycle, but for $12^\circ < \alpha < 17^\circ$ on the reducing incidence passage errors in C_L of up to 4% are observed. However, the present author does not consider the the additional accuracy arising from a further reduction in y sufficiently important, when taking the significant increase in computational costs into account associated with employing a mesh with a finer wall normal resolution.

Moreover, the discrepancies between the numerically and experimentally derived blade forces for the $\alpha = 13^\circ \pm 4^\circ$ tests are significantly greater than the 4% error in C_L between the results for Mesh I and Mesh VI_a, see Figure 3.11. Hence, it was determined to use Mesh VI_a for all further simulations conducted at $Re_c = O(10^6)$.

Chapter 4

Turbines at low blockage - Solidity study

This chapter presents a numerical investigation of the influence of turbine solidity on the hydrodynamics of a generic tidal cross-flow turbine; the dependency of the flow field and turbine performance on solidity are examined at two Reynolds Numbers, $Re_c = 4.42 \times 10^5$ & 2×10^6 ; Re_c is the mean blade Reynolds number around the azimuth and is given by:

$$Re_c = \frac{\rho c U_\theta}{\mu} \quad (4.0.1)$$

where ρ , μ , c and U_θ are respectively the fluid density, dynamic viscosity, blade chord and blade rotational velocity.

A number of developers of cross-flow turbines have carried out large scale laboratory experiments, for example see Gorlov (2003) and McAdam et al. (2010), and to further the understanding of such devices it is advantageous to conduct numerical simulations under similar conditions, i.e. $Re_c = O(10^5)$. Hence, it was determined to simulate flows through turbines at $Re_c = 4.42 \times 10^5$.

In addition, the numerical tests were repeated at $Re_c = 2 \times 10^6$ in order to examine the effect of changes in Re to a more realistic full scale value on the flow field and turbine performance.

The numerical investigation focuses on the computations of the two-dimensional (2D) flow field and turbine performance of a Darrieus-type turbine configuration tested at the Sandia National Laboratories (SNL), see Sheldahl and Klimas (1981); Table 4.1 shows the turbine’s parameters.

Number of blades	B	2
Hydrofoil		NACA 0015
Chord length	c	0.15m
Rotor radius	R	2.5m
Turbine solidity	σ	0.019
Rotor Height		5m

Table 4.1: Parameters of a turbine tested at the Sandia National Laboratories

In order to examine the influence of solidity, σ , on turbine performance three turbines were tested. To avoid the effects due to different blade Re , the chord length was kept constant, but the number of blades was varied from 2 to 3 to 4, resulting in turbine solidities of 0.019, 0.029 and 0.038 respectively, where:

$$\sigma = \frac{cB}{2\pi R} \quad (4.0.2)$$

where B , c and R are the number of blades, the blade chord length and the turbine radius respectively.

The computational domain, a 2D slice orthogonal to the turbine’s axis of rotation, is made up of three sub-domains, as illustrated in Figure 4.1:

1. a far-field domain shown in orange,
2. a turbine domain consisting of a circular rotating mesh, shown in green, and

3. discrete circular domains around each blade, shown in red.

The turbine is of radius $R = 16.67 c$. The domain extends $8 R$ upstream and $22 R$ downstream of the centre of the turbine and $8 R$ laterally to either side of the turbine centre. It is noted that the domain has modest blockage; $b = 12.5\%$, where blockage, b , is the ratio of the diameter of the turbine to the width of the domain.

Because the computational domain is 2D, the turbine blades are implicitly assumed to be infinitely long. A consequence of this assumption is that drag due to end plates or other blade support mechanisms is assumed negligible.

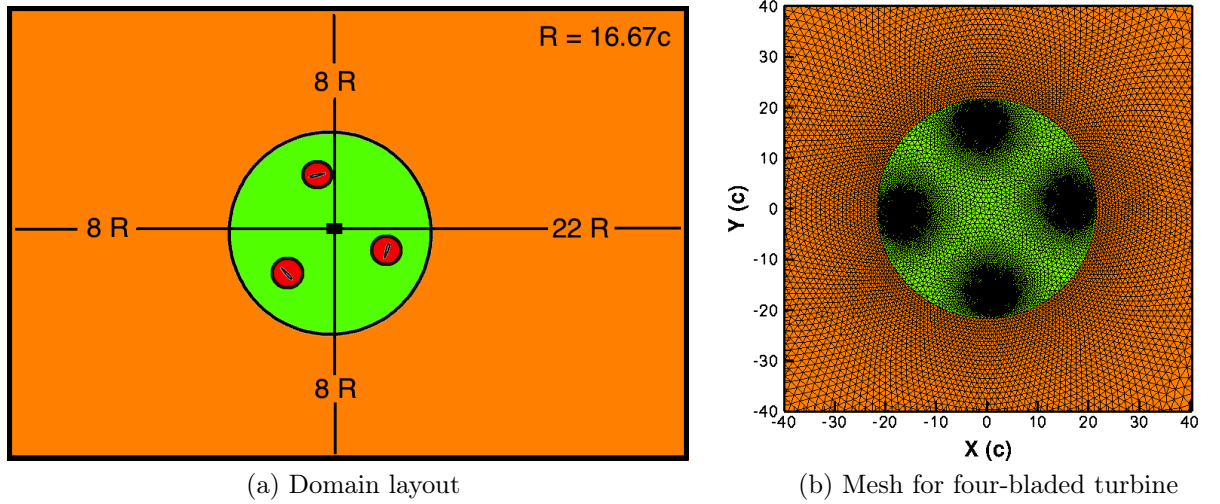


Figure 4.1: Computational domain for the present turbine solidity investigation

The following boundary conditions were employed in the simulations:

1. no slip condition on the blade surfaces,
2. unperturbed streamwise flow conditions across the upstream (inflow) and cross-stream boundaries, i.e. $U_x = 1 \text{ m/s}$ and $U_y = 0 \text{ m/s}$ and
3. a constant pressure condition along the downstream (outflow) boundary, i.e. $P = 0 \text{ Pa}$.

where U_x and U_y are the streamwise and cross-stream velocities respectively.

To simulate the rotation of the rotor, the circular turbine mesh with embedded blades is prescribed to move relative to the outer inertially fixed domain. As to the model discretisation, the PISO scheme was employed for pressure-velocity coupling, second-order upwind for the momentum as well as for the turbulence modelling equations.

4.1 Reynolds number = $O(10^5)$

The grid used for the discrete circular domains around each blade was a hybrid mesh composed of a very fine structured grid (long thin rectangular elements) for the near-wall region and unstructured triangular cells for the remainder of the subdomain; the mesh employed for the simulations conducted at $Re_c = 4.42 \times 10^5$ was Mesh 2, which was considered most appropriate following the extensive validation presented in Section 3.1.

For Mesh 2, the first grid spacing from the surface in the normal direction was $1.45 \times 10^{-4} c$, which resulted in $1 < y^+ < 5$ depending on blade incidence. With a streamwise mesh resolution of 744 cells per blade side this resulted in 94,100 cells for the circular domain around each blade and in a total number of elements in the entire turbine domain of 426,000 for the 2-bladed, 531,400 for the 3-bladed and 631,900 for the 4-bladed turbine configurations.

4.1.1 Solution convergence

Solution convergence was determined by examining the torque coefficient, C_m , for a single blade, and the power coefficient, C_P , for the full turbine, see Figure 4.2, 4.5 & 4.6. Moreover, the development of the wake was studied by examining the centreline wake velocity magnitude as well as the velocity magnitude at the cross-stream wake traverses at 1 diameter, D , 5 D and 10 D downstream of the turbine, see Figure 4.3,

4.4 & 4.7.

A simulation was considered statistically converged, when, for 10 consecutive revolutions, turbine C_P did not change to the second significant figure and the various wake velocity plots remained unaltered for 5 consecutive revolutions. Figure 4.2 - 4.7 show that the number of revolutions required to attain a statistically converged solution varied with turbine solidity, σ ; the higher σ , the more revolutions were required to achieve solution convergence. For all three turbine configurations, an example of a detailed analysis that was carried out to determine solution convergence is given below, at $\lambda = 3$.

From Figure 4.2b it is apparent that C_P remained unaltered to the second significant figure after 8 revolutions for the 2-bladed turbine at $\lambda = 3$, but some of the wake velocities were still varying cycle-to-cycle.

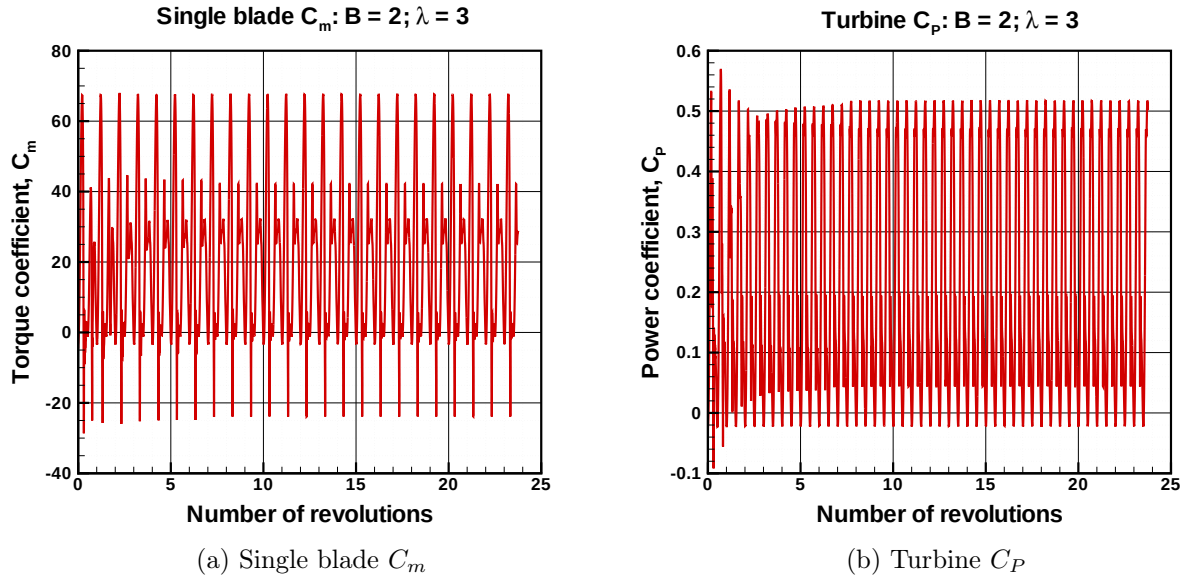
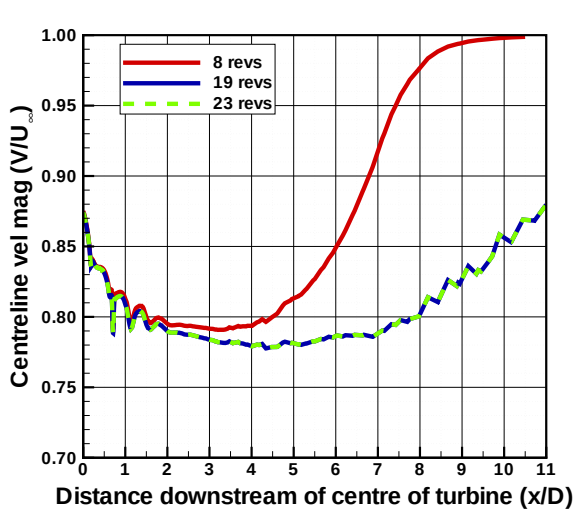


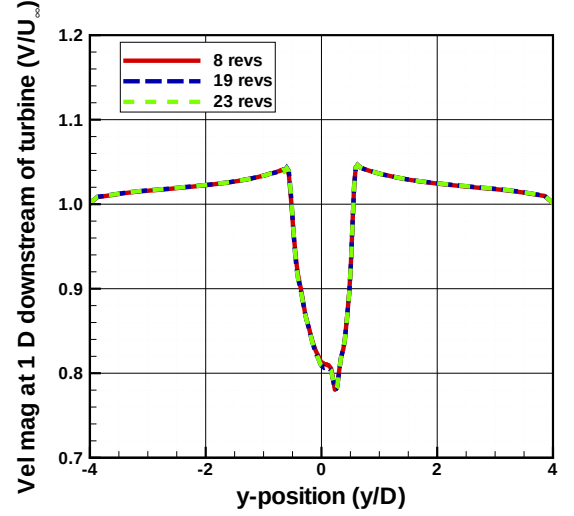
Figure 4.2: Convergence of blade torque and turbine power histories for $B = 2$, $\sigma = 0.019$ and $\lambda = 3$

Figure 4.3b illustrates that the velocity magnitude at the cross-stream traverse at $1D$ downstream of the turbine remained unaltered after revolution 8, but the flow field

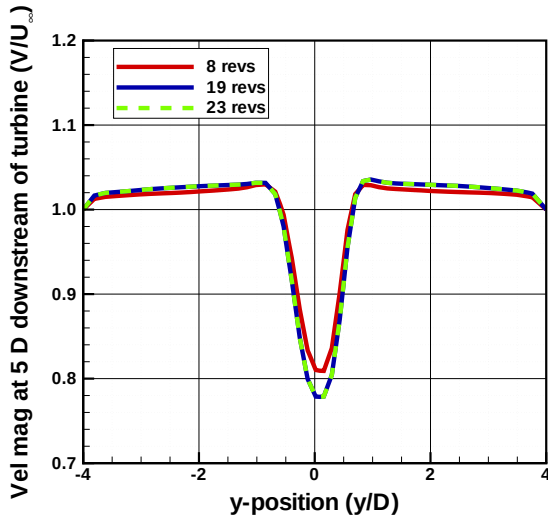
was not considered fully converged until revolution 19, as both the centreline velocity magnitude as well as the velocity magnitude at the cross-stream traverse at $10 D$ downstream of the turbine were still changing cycle-to-cycle until revolution 19, see Figure 4.3a & 4.3d. Due to wake convection convergence takes longer, as the flow field is examined further downstream.



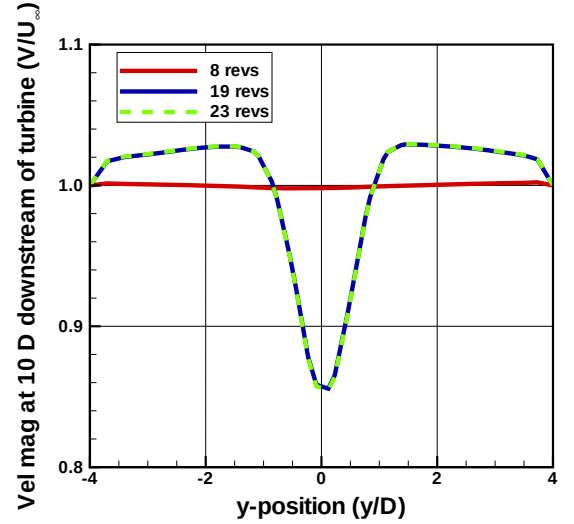
(a) Centreline velocity magnitude



(b) Velocity magnitude for a cross-stream traverse at $1 D$ downstream of turbine



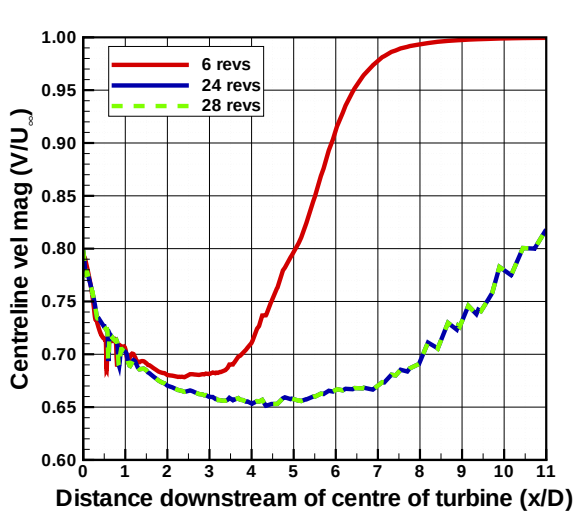
(c) Velocity magnitude for a cross-stream traverse at $5 D$ downstream of turbine



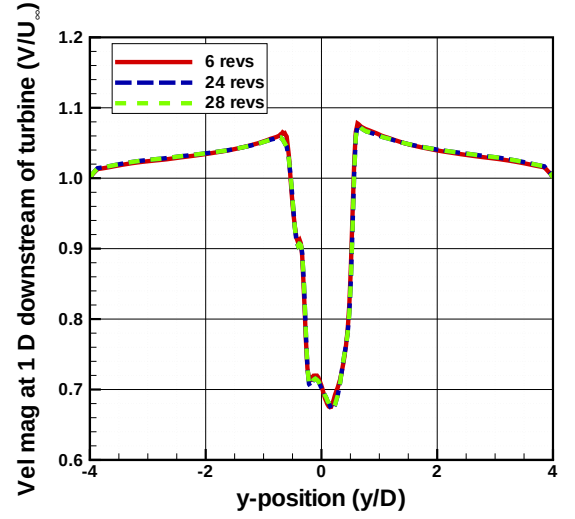
(d) Velocity magnitude for a cross-stream traverse at $10 D$ downstream of turbine

Figure 4.3: Wake velocity profiles for $B = 2$, $\sigma = 0.019$ and $\lambda = 3$

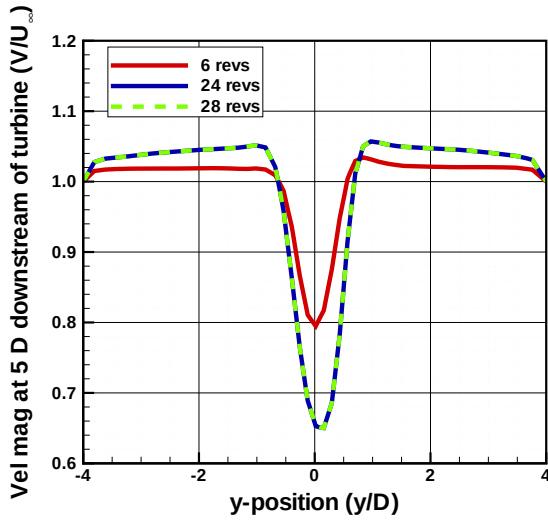
For the 3-bladed turbine at $\lambda = 3$, it took 5 more revolutions than for the 2-bladed configuration until a converged solution was attained, see Figure 4.4 & 4.5 . While the wake velocities did not remain unaltered until after revolution 24, see Figure 4.4, C_P reached a constant value after revolution 8 for $\sigma = 0.029$ ($B = 3$), see Figure 4.5.



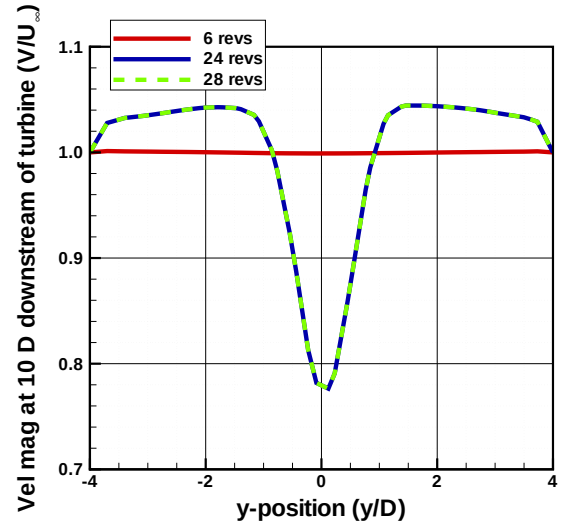
(a) Centreline velocity magnitude



(b) Velocity magnitude for a cross-stream traverse at 1 D downstream of turbine



(c) Velocity magnitude for a cross-stream traverse at 5 D downstream of turbine



(d) Velocity magnitude for a cross-stream traverse at 10 D downstream of turbine

Figure 4.4: Wake velocity profiles for $B = 3$, $\sigma = 0.029$ and $\lambda = 3$

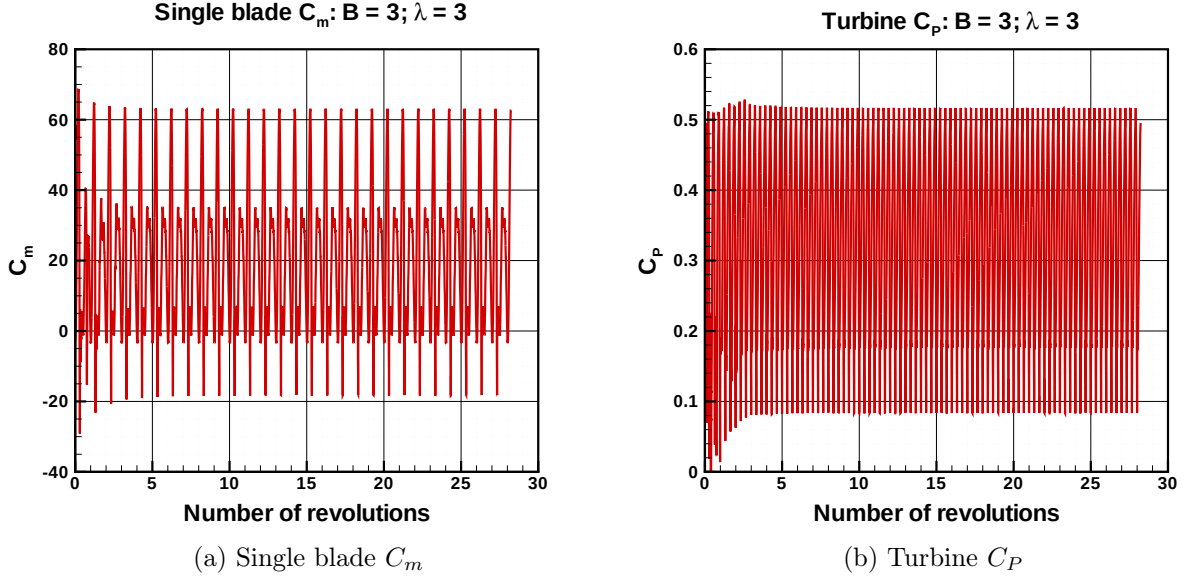


Figure 4.5: Convergence of blade torque and turbine power histories for $B = 3$, $\sigma = 0.029$ and $\lambda = 3$

As to the simulations for $\sigma = 0.038$ ($B = 4$) at $\lambda = 3$, the solution was considered statistically converged after revolution 27, see Figure 4.7.

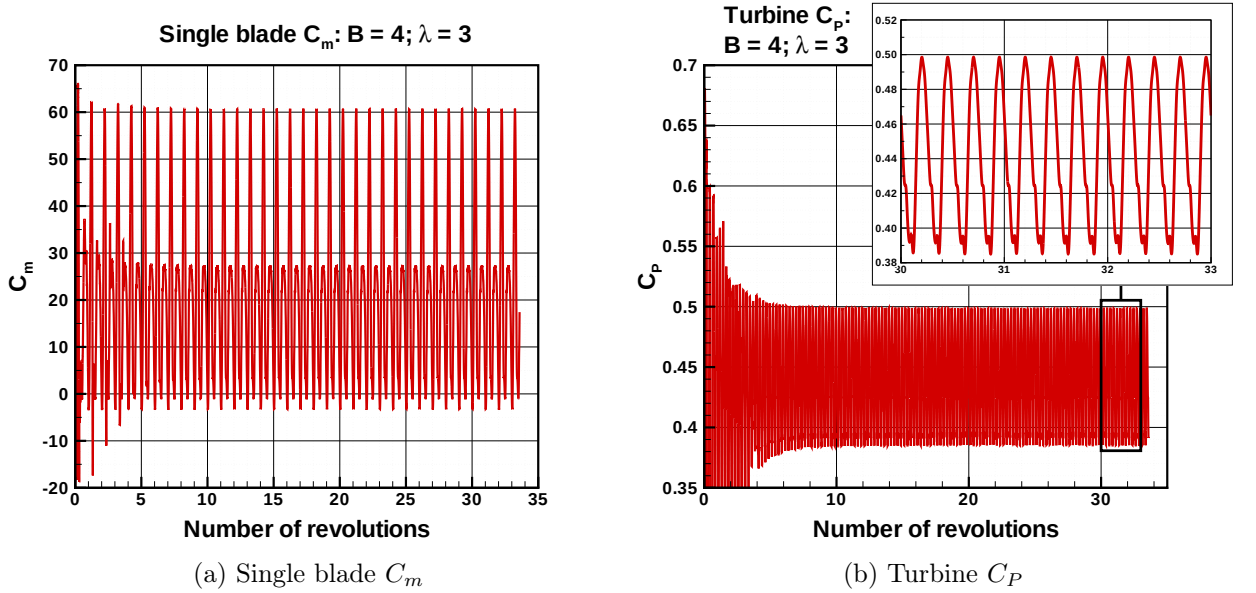
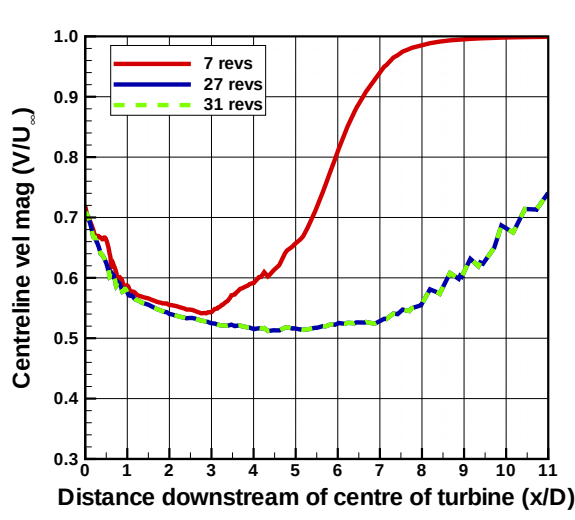
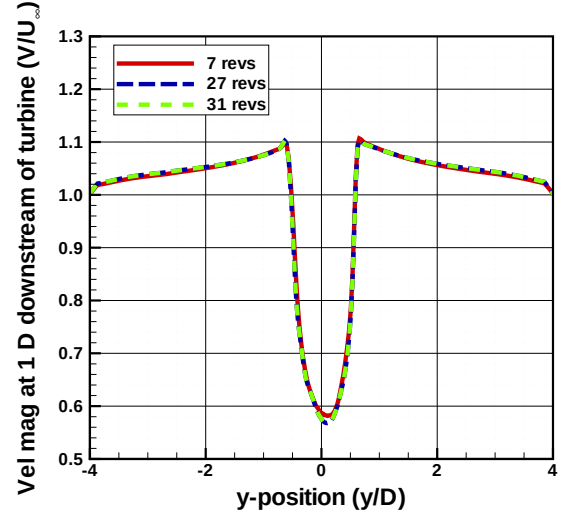


Figure 4.6: Convergence of blade torque and turbine power histories for $B = 4$, $\sigma = 0.038$ and $\lambda = 3$

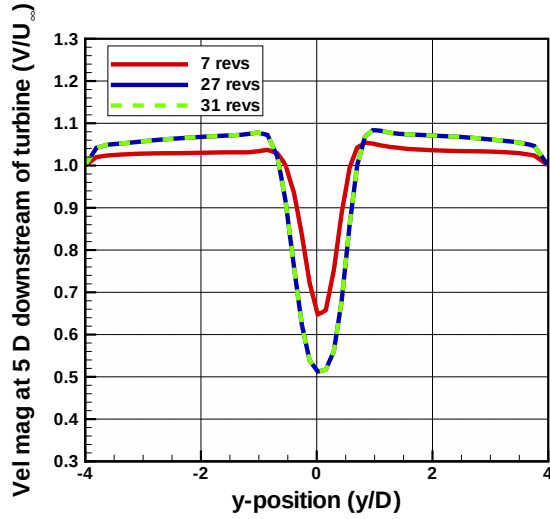
As for the 2- and 3-bladed turbines, when examining the blade forces, convergence was reached earlier; C_P did not change to the second significant figure after revolution 13, but it took 14 more revolutions until the various wave velocities investigated remained unaltered.



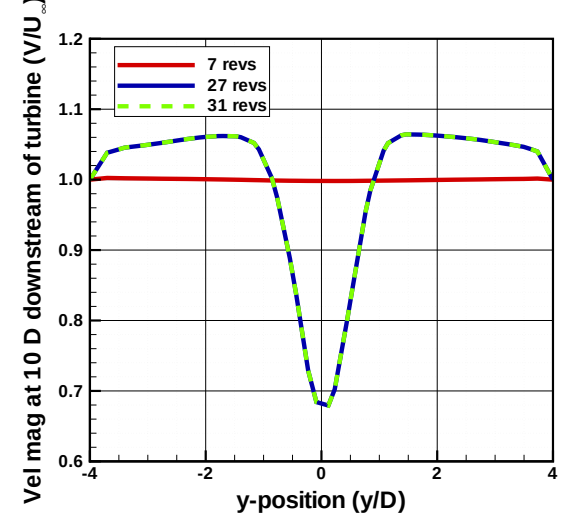
(a) Centreline velocity magnitude



(b) Velocity magnitude for a cross-stream traverse at 1 D downstream of turbine



(c) Velocity magnitude for a cross-stream traverse at 5 D downstream of turbine



(d) Velocity magnitude for a cross-stream traverse at 10 D downstream of turbine

Figure 4.7: Wake velocity profiles for $B = 4$, $\sigma = 0.038$ and $\lambda = 3$

Moreover, in Figure 4.7b - 4.7d asymmetries may be observed for the velocity magnitude at the cross-stream traverses after revolution 7, but as the solution reaches convergence these disappear. These asymmetries are associated with a starting vortex which is washed downstream over time, rendering a more symmetric steady-state wake profile.

Figure 4.8 shows the number of revolutions required until C_P remained unchanged for 10 consecutive revolutions, for a range of tip speed ratios simulated for the three different turbine solidities.

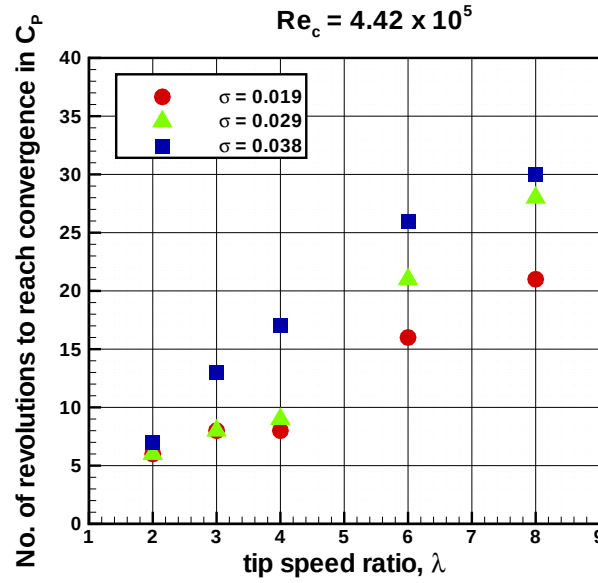


Figure 4.8: Number of revolutions required for convergence in C_P

From Figure 4.8 it is apparent that the number of revolutions required to reach convergence increases, as turbine solidity increases. Moreover, it may be noted that the larger λ , the more revolutions are required until solution convergence is reached. Both observations may be expected, as both a decrease in the rotational velocity, ω , associated with a decrease in λ as well as a reduction in the streamwise flow velocity induced by an increase in solidity, mean that a larger number of revolutions is required for free convection of a particle through the entire domain in the streamwise direction.

Furthermore, it is noted that an increase in thrust results in a bigger impact on the

flow field, i.e. a greater perturbation on the flow field, and hence the longer it takes to reach convergence in the near field. As discussed below, both an increase in turbine solidity and tip speed ratio lead to an increase in thrust and hence to an increase in the time required to reach convergence.

4.1.2 Results

Figure 4.9 shows the power and thrust curves for all three turbines, where $\sigma = 0.019$, 0.029 and 0.038, operating at $Re_c = 4.42 \times 10^5$; the power and thrust coefficients are defined as follows:

$$C_P = \frac{Power_{mech}}{Power_{kinetic}} = \frac{Q\omega}{\frac{1}{2}\rho U_\infty^3 A} \quad (4.1.1)$$

$$C_T = \frac{T}{\frac{1}{2}\rho U_\infty^2 A} \quad (4.1.2)$$

where Q , T , A are respectively the resultant turbine torque per unit span, turbine thrust per unit span and the projected frontal area of the turbine per unit span of turbine, i.e. $A = 2R$.

For the present simulations the tip speed ratio, λ , defined in Equation 4.1.3, was altered by changing the turbine angular velocity, ω , and Re_c was maintained by adjusting μ accordingly.

$$\lambda = \frac{U_\theta}{U_\infty} = \frac{\omega R}{U_\infty} \quad (4.1.3)$$

where U_∞ is the free-stream velocity.

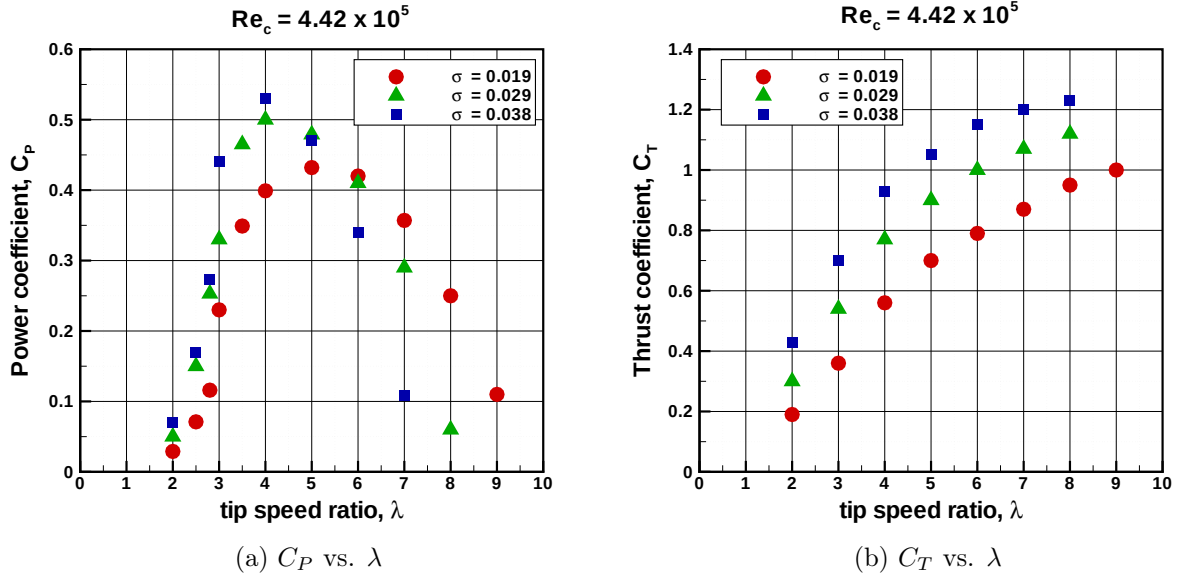


Figure 4.9: Power and thrust coefficient variation for varying turbine solidity, σ , at $Re_c = 4.42 \times 10^5$

A typical power curve for cross-flow turbines takes a bell shape, as shown in Figure 4.9a. This is, as can be deduced from Figure 1.12b, because at low tip speed ratio, λ , the flow incidence can be high and thus torque and power are limited by blade stalling, whilst the limiting factor at high λ is low lift due to low blade incidence, α ; peak power takes place in between, thus leading to a bell shaped power curve.

The thrust curves also exhibit the expected shape. Firstly, the thrust coefficient, C_T , increases, as tip speed ratio, λ , is increased and secondly, C_T becomes larger, the higher turbine solidity, in this case the more blades the turbine comprises.

When comparing the different power curves, a number of differences may be observed:

1. $\max C_P$ is 26% larger for the 4-bladed turbine than for the 2-bladed machine;
2. a narrowing and shift of the power curve to a lower range of λ , as solidity is increased.

These differences can be considered by analysing the influence of turbine solidity on the streamwise velocity.

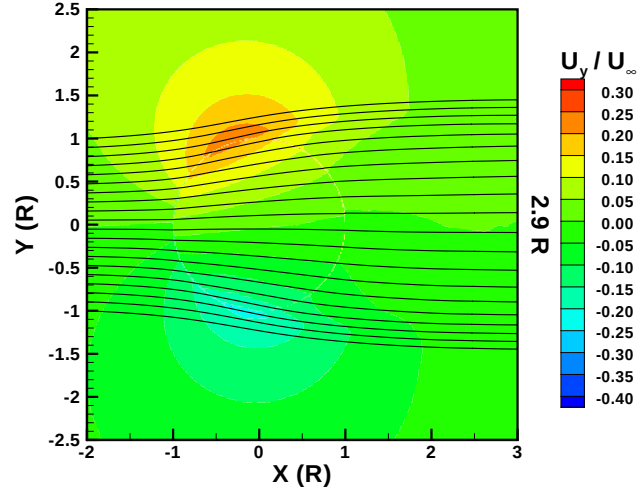
The optimum design configuration lies between two extremes; a turbine that presents a very high and a very low impedance. On the one hand, the power output is proportional to the thrust that the turbine exerts on the flow. However, the larger the thrust the larger the flow impedance and hence the lower the flow velocity and kinetic energy flux through the turbine. In order that the turbine presents an “efficient” impedance to the flow, the right balance of solidity and tip speed ratio is sought. In essence, the higher solidity is, the lower λ will be for max power removal.

4.1.2.1 Time-averaged flow fields

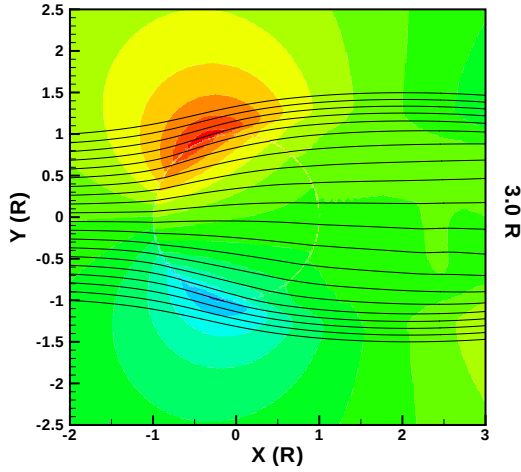
Figure 4.10 shows the time-averaged flow fields for the three different turbine configurations for $\lambda = 8$ in each case. The colour shading in these plots depicts the cross-stream velocity component, U_y . The time-averaged flow fields were obtained by:

1. generating flow fields at azimuthal increments of 5 degrees, i.e. 72 different flow fields per revolution, for a converged turbine cycle,
2. projecting each flow field to a rectangular stencil with a resolution of $0.25\ c$ in both the streamwise and cross-stream directions,
3. extracting the streamwise and cross-stream velocities, where the velocities at the position of the blades were replaced with the blade rotational velocity, U_θ , and
4. taking a time-average of the 72 different turbine azimuthal positions.

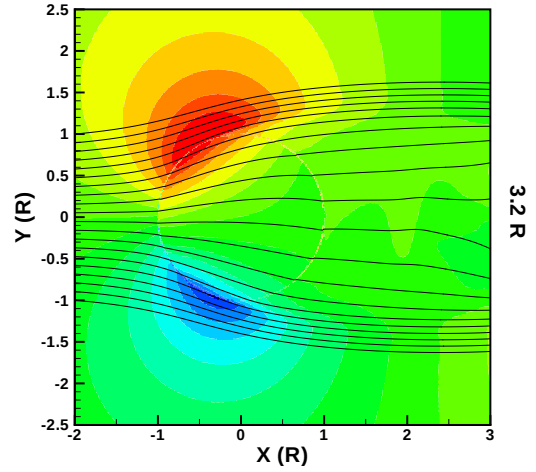
The same procedure was followed for increments of 2 degrees over up to 3 revolutions, but neither the increase in the number of revolutions over which the average was taken nor the increase in the number of azimuthal positions per revolution had an influence on the solution. Moreover, convergence tests were conducted for the stencil resolution.



(a) $\sigma = 0.019$, $\lambda = 8$, $C_P = 0.25$



(b) $\sigma = 0.029$, $\lambda = 8$, $C_P = 0.06$



(c) $\sigma = 0.038$, $\lambda = 8$, $C_P = -0.19$

Figure 4.10: Time-averaged flow fields at $\lambda = 8$ for three different solidities; $\sigma = 0.019$, 0.029 & 0.038 , together with instantaneous streamlines

On the three time-averaged velocity plots streamlines have been added in order to facilitate the comparison of the flow impedance for each case. The size of all streamtubes is set to be $2 R$, at a distance $2 R$ upstream of the centre of the turbine. By comparison of the expansion of the streamtubes, it is evident that, as expected, the higher the solidity, the larger the flow impedance. This is re-iterated by the results for the thrust coefficient, C_T , plotted in Figure 4.9b; C_T is 30% larger for the 4-bladed turbine than

the 2-bladed machine at $\lambda = 8$.

Correspondingly, by continuity, the cross-stream velocity component, U_y , is observed to increase with solidity.

As mentioned above, a change in the streamwise velocity affects the kinetic energy flux through the turbine. Furthermore, it influences the range of incidences the blades are presented with throughout the revolution cycle, with consequence to the power generated at different tip speed ratios. Below we first consider the influence of solidity, σ , on high λ power generation and then the effect on low λ power generation.

Figure 4.9a shows that at high λ , the higher the impedance, the lower the performance of the turbine. This is because at high tip speed ratios the performance of the turbine is limited by low angles of attack, which result in low lift and thus low torque. As a reduction in the streamwise flow velocity acts to further reduce the angle of attack, an increase in solidity leading to an increase in flow impedance has an adverse effect on turbine performance at high tip speed ratios. In addition, a reduction in the streamwise flow velocity reduces the kinetic energy flux through the turbine.

However, the effect of increasing turbine solidity, σ , reverses at low tip speed ratios; see Figure 4.11.

As the relative expansion of the streamtube indicates, at $\lambda = 3$ an increase in σ also leads to an increase in flow impedance. Again, this is supported by comparison of the thrust coefficients in Figure 4.9b, which underlines for $\lambda = 3$ that the larger σ , the higher C_T , and thus flow impedance.

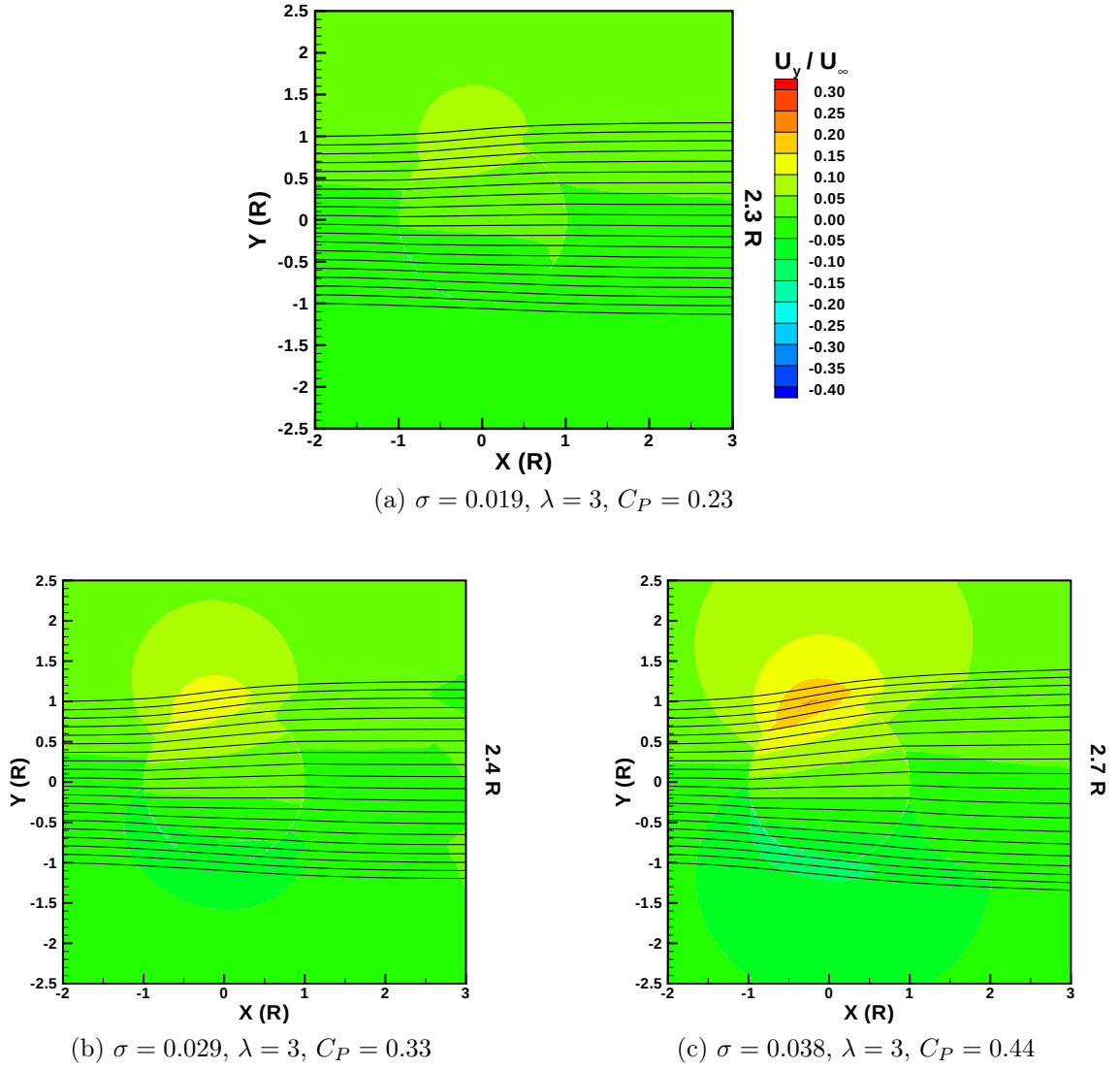
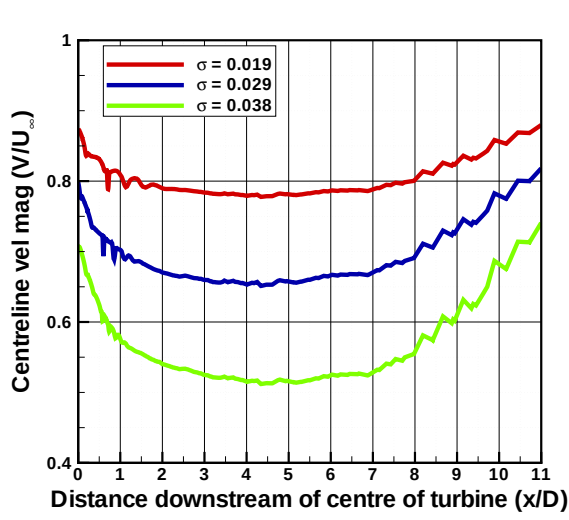
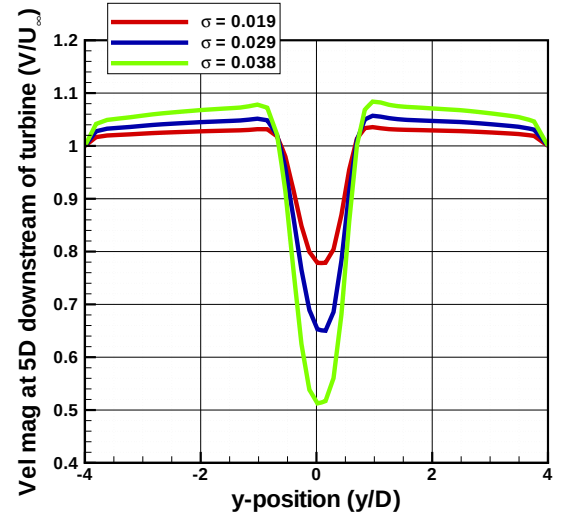


Figure 4.11: Time-averaged flow fields at $\lambda = 3$ for three different solidities; $\sigma = 0.019$, 0.029 & 0.038, together with instantaneous streamlines

Also, a comparison of the velocity magnitude along the centreline as well as a cross-stream traverse $5 D$ downstream of the turbine show that the higher σ , the lower the velocity through the turbine as well as in the turbine's wake, see Figure 4.12. It may be noted from Figure 4.12b that the flow velocity magnitude for $|y| \gtrsim 0.5 D$ increases with an increase in turbine solidity, which is necessary to satisfy continuity in a domain of finite width (however large that may be).



(a) Centreline velocity magnitude plot



(b) Velocity magnitude plot for 5 D downstream of turbine

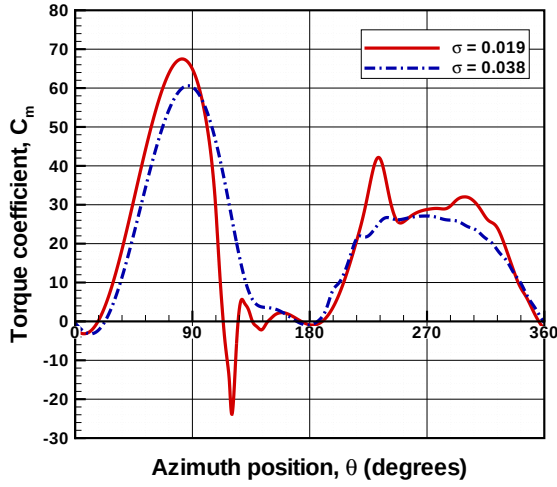
Figure 4.12: Velocity magnitude comparison at $\lambda = 3$ for three turbine solidities; $\sigma = 0.019, 0.029$ & 0.038

However, in contrast to the high tip speed ratio case, where low angles of attack limit the performance of the turbine, at low tip speed ratios high angles of attack leading to flow separation are the limiting factor. Hence, an increase in flow resistance resulting in lower angles of attack may lead to an increase in power despite a reduction of the available kinetic energy flux flowing through the turbine due to a reduction in the streamwise flow velocity; for instance, when comparing power coefficients, C_P , at $\lambda = 3$ for 2-, 3- and 4-bladed turbines, the higher solidity machine produces 91% and 33% more power than the 2- and 3-bladed devices respectively.

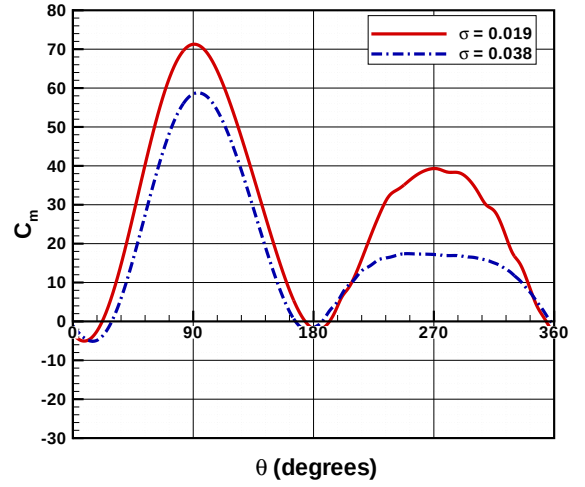
However, the significant difference in performance between the 2- and 4-bladed turbines cannot be fully explained by a reduction in angle of attack associated with the increase in flow impedance. The analysis of this particular case is more complex and will be addressed below.

4.1.2.2 Blade torque

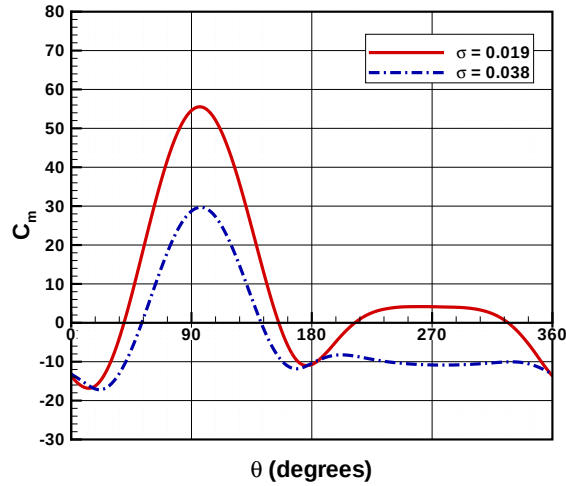
In order to enable a more detailed analysis, the performance of an individual blade of the 2- and 4-bladed turbines has been studied at various λ . Figure 4.13 shows the torque trace of an individual blade of the 2- and 4-bladed turbines operating at $\lambda = 3$, 4 & 8.



(a) Blade C_m traces at $\lambda = 3$



(b) Blade C_m traces at $\lambda = 4$



(c) Blade C_m traces at $\lambda = 8$

Figure 4.13: Comparisons of blade torque coefficient, C_m , histories of the 2- and 4-bladed turbines

Typically, the torque history of a blade for one complete cycle, as shown in Figure 4.13a - 4.13c, has two peaks, which correspond to the azimuth positions, θ , where α reaches its maximum pre-stall angle. In the absence of blade stall these positions correspond to azimuth angles of $\theta \approx 90^\circ$ and $\theta \approx 270^\circ$, measured anti-clockwise from the top of the turbine, assuming negligible effects of velocity induction. On examination of Figure 4.13a - 4.13c the torque peaks are indeed found to lie close to $\theta = 90^\circ$ and $\theta = 270^\circ$.

However, a significant difference in the maximum torque may be noted between the upstream and downstream blade passes. The large reduction in max torque on the downstream passage is due to the streamwise flow impedance presented by the turbine. In fact, the higher λ or σ , the larger the reduction in U_x through the turbine, which results in:

1. a larger reduction in the kinetic energy flux passing through the downstream passage and
2. lower incidences the blades are presented with on the downstream passage.

This is why the (positive) torque contributed by the downstream passage is lowest for the 4-bladed turbine operating at $\lambda = 8$ out of all the cases shown in Figure 4.13; in fact, for $\sigma = 0.038$ & $\lambda = 8$, the torque generated by a blade for $180^\circ < \theta < 360^\circ$ is negative, see Figure 4.13c.

In Figure 4.13a - 4.13c, the torque contribution per blade has only been considered. Essential with regard to the turbine's performance is whether the deficit in power generated per blade per cycle for the higher solidity turbine is compensated for by the torque provided by the additional blades.

At $\lambda = 8$ the adverse effects of the increase in solidity outweigh the additional torque due to the extra blades, and the 4-bladed turbine produces less, in fact negative, power than the 2- and 3-bladed turbines, see Figure 4.9a.

In contrast, at moderate tip speed ratios, at $\lambda = 4$ say, the additional torque provided by the additional blades of the higher solidity device offsets the reduction in net torque per blade such that at $\lambda = 4$ the higher solidity device yields more power, see Figure 4.9a.

At lower tip speed ratios, increasing solidity may further increase the turbine's power take-off relative to the lower solidity device, although the mechanism can be more complex. In Figure 4.13a it is seen that the blade torque of the 2-bladed turbine experiences significant fluctuations, which are associated with large scale vortex shedding that occur as a result of flow separation. The net result is that there is little change in net torque per cycle per blade between the two 2- and 4-bladed turbine configurations, but an increase in total turbine torque for the higher solidity device, as there are more blades.

4.1.2.3 Instantaneous streamline plots

The differences between the two turbine configurations at $\lambda = 3$ are underlined by comparison of instantaneous streamline plots shown in Figure 4.14.

Figure 4.14a - 4.14d show streamline plots for the 2-bladed turbine operating at $\lambda = 3$ for four different blade azimuth positions over the upstream half of the turbine. It is evident that the blades of the lower solidity turbine operate under stalled conditions for $90^\circ < \theta < 160^\circ$, which is the cause of the low blade torque generated for $110^\circ < \theta < 160^\circ$, see Figure 4.13a. The flow is observed to reattach at around $\theta = 160^\circ$.

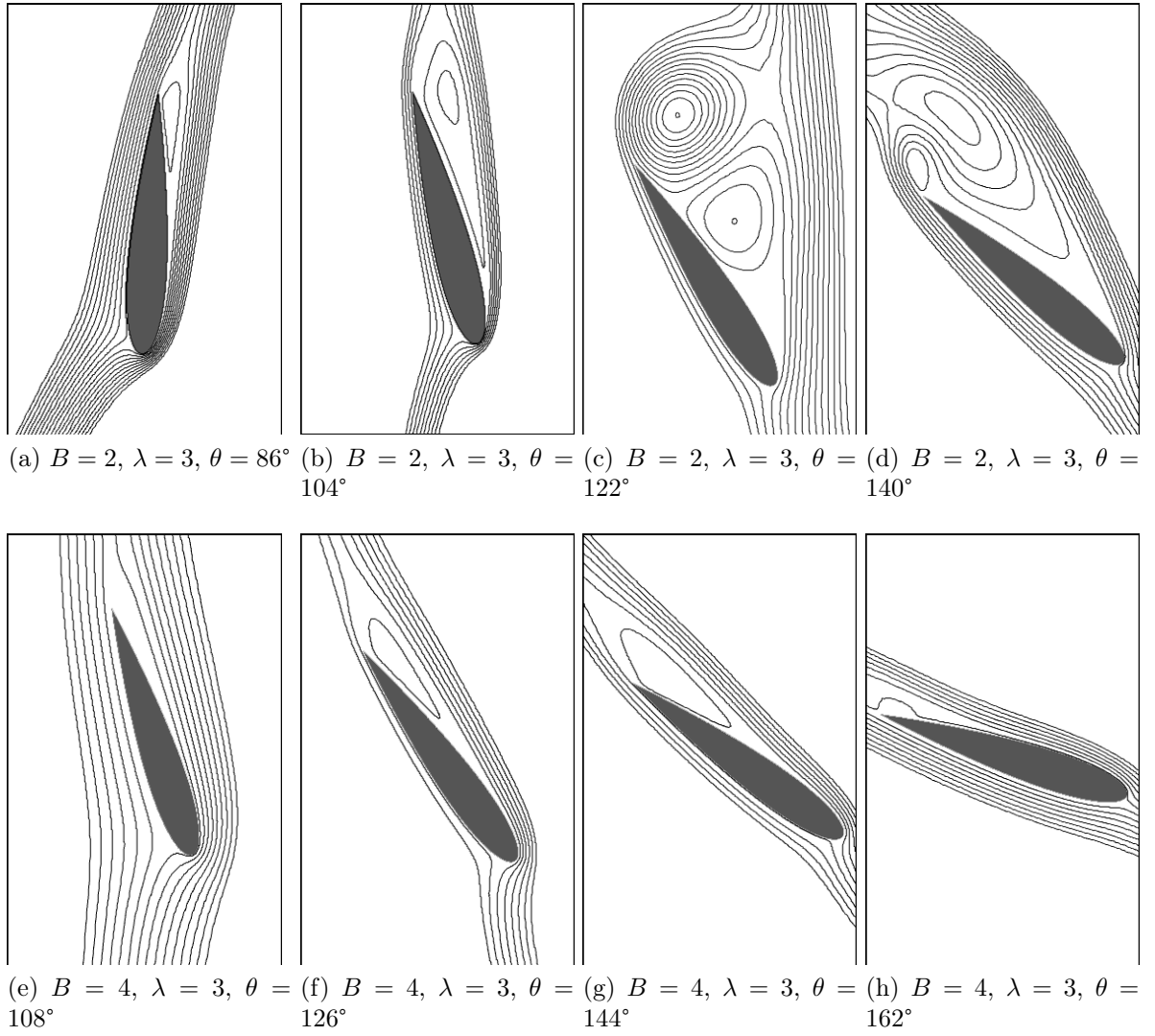
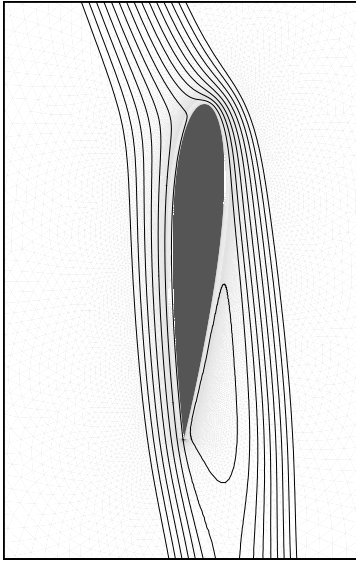


Figure 4.14: Instantaneous streamline plots for the 2- & 4-bladed turbines for $86^\circ < \theta < 162^\circ$

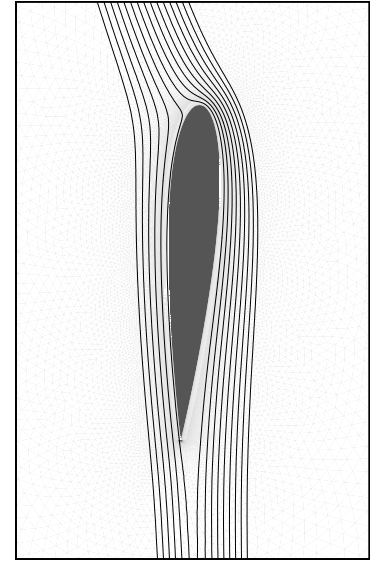
Figure 4.14e - 4.14h show streamline plots for the 4-bladed turbine operating at $\lambda = 3$ for four different blade positions, θ , over the second quadrant of revolution. It is apparent that the flow over the blades of the higher solidity turbine starts to separate at the trailing edge, and that the point of separation moves towards the leading edge with increasing azimuth angle. However, unlike the lower solidity device, the 4-bladed turbine's blades do not undergo full separation and do not initiate gross scale vortex shedding. The more attached behaviour of the higher solidity device is due to the

maximum angle of attack being lower than for the lower solidity machine due to the increased flow impedance of the 4-bladed configuration. Hence, the 4-bladed turbine exhibits superior performance for $110^\circ < \theta < 160^\circ$, see Figure 4.13a.

On the downstream passage of the turbine, the flow over the blades of the 2-bladed turbine operating at $\lambda = 3$ was observed to separate from the blade's trailing edge at around $\theta = 240^\circ$ and a large vortex structure formed by $\theta = 266^\circ$, see Figure 4.15a, whilst for the blades of the 4-bladed turbine a more attached behaviour was observed on the downstream pass, see Figure 4.15b; this is due to the lower incidences reached that can be attributed to the increase in flow impedance for the higher solidity turbine.



(a) $\lambda = 3$, $B = 2$, $\theta = 266^\circ$



(b) $\lambda = 3$, $B = 4$, $\theta = 270^\circ$

Figure 4.15: Instantaneous streamline plots for the 2- & 4-bladed turbines for $\theta \approx 270^\circ$

However, despite the extensive region of separated flow the hydrofoil of the 2-bladed turbine generates significantly higher torque than its 4-bladed counterpart for $210^\circ < \theta < 330^\circ$, see Figure 4.13a. The torque peaks observed for the 2-bladed turbine on the downstream passage are a result of increased lift force on blades operating beyond static stall conditions and are therefore a result of dynamic effects.

Also, it is important to note that the RANS assumption, see Section 2.2.1, is valid for flows, which include large scale unsteady coherent structures, as long as the time period over which the turbulence is being averaged is significantly smaller than the eddy shedding time scales. With regard to the simulations discussed above for the 2- & 4-bladed turbines operating at $\lambda = 3$, the time period of the vortex shedding, which is half the time period of one turbine revolution, is about 2 orders of magnitude larger than the time period of the averaging process, i.e. the time step employed, $\Delta t = 5 \times 10^{-3}$ seconds. Hence, the RANS assumption is still valid for the present simulations.

4.1.2.4 Sectional lift and drag forces - indication of dynamic stall

From the time-mean flow fields, it was possible to determine the lift and drag coefficients, C_L and C_D , as well as the angle of incidence, α . In doing this care was taken to neglect the highly perturbed flow field adjacent to the blades. This was achieved by neglecting a thin annular region that encloses the blade orbit. This was facilitated through the use of a polar sampling grid. As before, an extensive convergence study was carried out to determine the optimal stencil resolution as well as the required annulus width. For the results presented below, a resolution of 5 degrees in θ , and $0.33 c$ in the radial direction was used and a ring width of $0.33 c$ at the blades' position.

Figure 4.16 & 4.17 show force and velocity diagrams respectively, which highlight how the forces were resolved to work out C_L and C_D and the velocities to work out α and the resultant velocity, U_r .

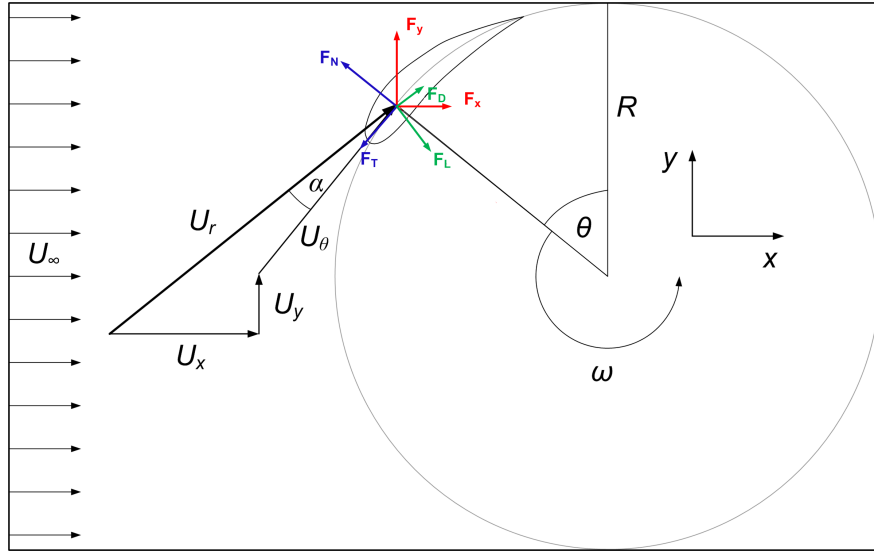


Figure 4.16: Force diagram - cross-flow turbine, adapted from McAdam (2008)

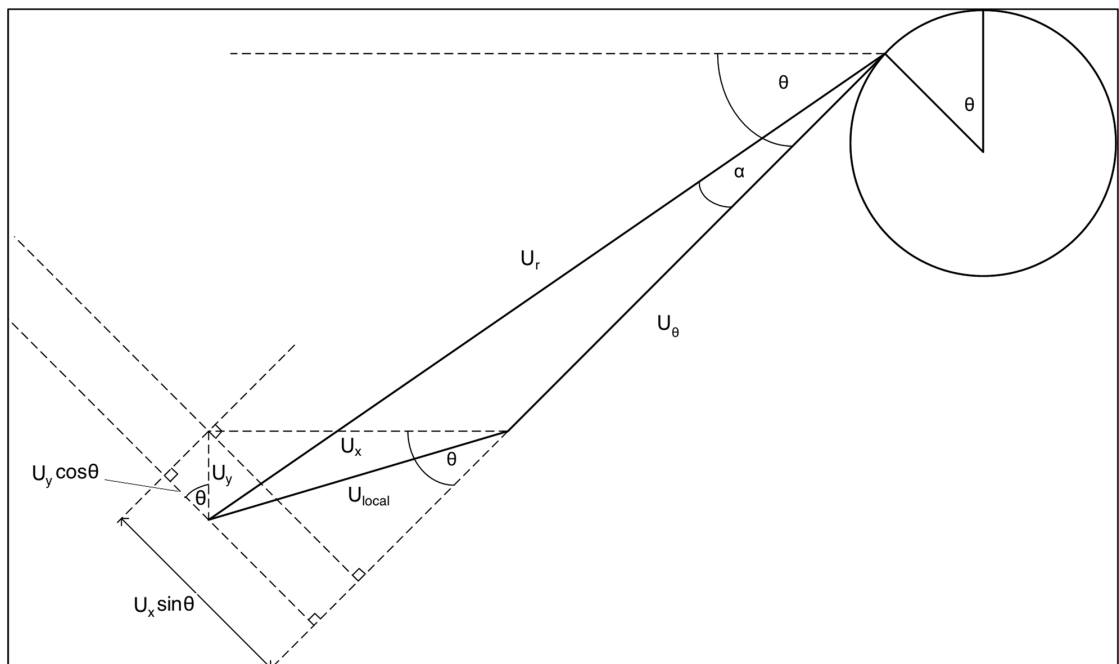


Figure 4.17: Velocity diagram - cross-flow turbine

The streamwise and cross-stream sectional blade forces, F_x & F_y , are resolved into normal and tangential sectional blade force coefficients, C_{F_N} & C_{F_T} :

$$C_{F_N} = \frac{1}{\frac{1}{2}\rho U_r^2 c} (F_y \cos \theta - F_x \sin \theta) \quad (4.1.4)$$

$$C_{F_T} = \frac{1}{\frac{1}{2}\rho U_r^2 c} (-F_y \sin \theta - F_x \cos \theta). \quad (4.1.5)$$

The resultant flow velocity, U_r , is given by:

$$U_r = \sqrt{(U_x + U_\theta \cos \theta)^2 + (U_y + U_\theta \sin \theta)^2} \quad (4.1.6)$$

$$U_r = \sqrt{(U_x \sin \theta - U_y \cos \theta)^2 + (U_\theta + U_x \cos \theta + U_y \sin \theta)^2} \quad (4.1.7)$$

where U_x & U_y are the local streamwise and cross-stream velocity components and U_θ the blade rotational velocity. Note that U_x & U_y include the effects of streamwise and torsional flow inductance on the free-stream flow.

Hence, the sectional blade lift and drag coefficients, C_L & C_D , can be determined:

$$C_L = -C_{F_N} \cos \alpha + C_{F_T} \sin \alpha \quad (4.1.8)$$

$$C_D = -C_{F_N} \sin \alpha - C_{F_T} \cos \alpha \quad (4.1.9)$$

where the angle of attack, α , is calculated from the time-averaged flow field as follows:

$$\alpha = \arctan \left(\frac{U_x \sin \theta - U_y \cos \theta}{U_\theta + U_x \cos \theta + U_y \sin \theta} \right) \quad (4.1.10)$$

Figure 4.18 shows the sectional lift and drag forces experienced by a single blade of the 2-bladed turbine at $\lambda = 3$, as it traverses a full rotation cycle. In the manner plotted, positive incidence refers to the downstream pass of the blade, whilst negative incidence refers to the upstream blade pass. The markers 1, 2, 3 and 4 indicate the azimuthal positions of the start of the 1st, 2nd, 3rd and 4th quadrants at $\theta = 0^\circ$, 90° , 180° and 270° respectively. Also, the azimuthal positions corresponding to $\alpha = 0^\circ$ have been marked.

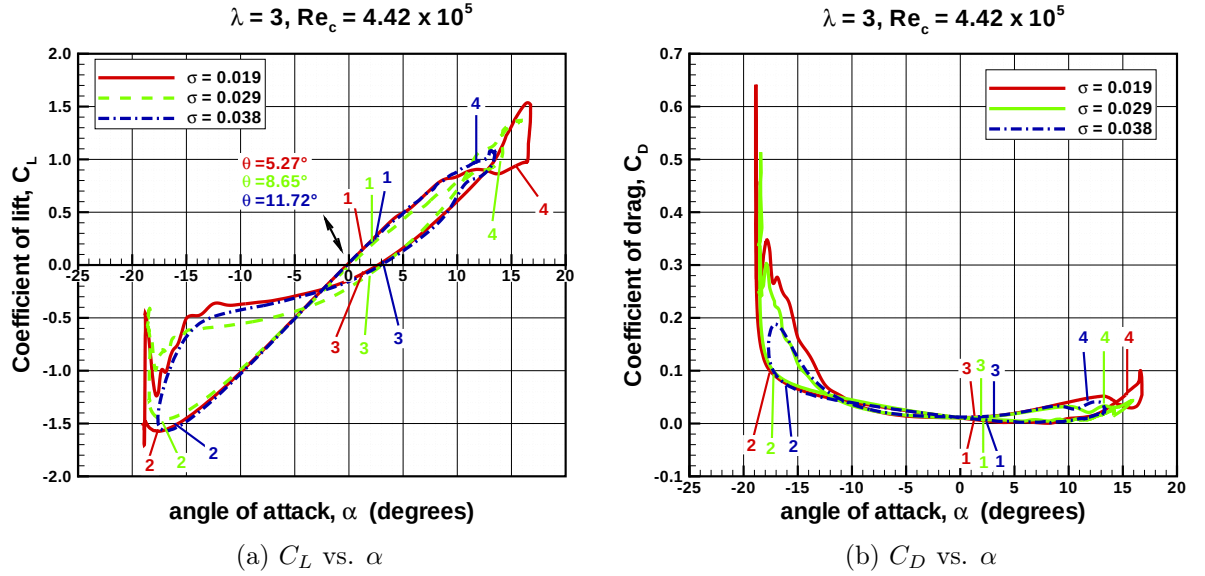


Figure 4.18: Blade coefficient histories for a blade of the 2-bladed turbine at $\lambda = 3$

Examining the blade's C_L and C_D variation, it is evident that the blades of the 2- and 3-bladed turbines experience dynamic stall on both the upstream and downstream passages of the turbine, similar to dynamic stall events reported in McCroskey (1981), Carr (1988) and Tang and Dowell (1995). For the 2-bladed turbine, for example, static stall, which is expected to occur at around $\alpha = 13^\circ$ at this Re_c , see Figure 3.1, is

exceeded by about 5.9° on the upstream and 3.8° on the downstream pass, and the blades experience lift and drag force hystereses typical of dynamic stall.

Dynamic stall is a non-linear unsteady hydrodynamic effect that can significantly influence a blade's performance. Dynamic stall requires a periodic variation in incidence, in which the maximum and minimum incidence exceed and fall below the static stall angle, α_{ss} , as discussed in McCroskey (1981). Provided the increase in α is rapid enough, the flow remains attached despite passing α_{ss} . Near the trailing edge the flow starts to reverse and moves up the chord, as described in Carr (1988). Once it covers most of the foil section, the leading edge flow no longer remains attached and a strong vortex-like disturbance develops, which spreads rearwards. This vortex induces a highly non-linear pressure field and hence distorts the pressure distribution along the blade's surface. This distortion allows lift to be generated past the static stall, the main advantage associated with dynamic stall.

However, it should also be noted that once stall does occur, it tends to be more severe than for the static case. Moreover, drag significantly increases during dynamic stall. A third negative effect is that reattachment takes place at an angle of attack below that of static stall, resulting in a hysteresis loop, such as those shown in Figure 4.18a.

Moreover, Figure 4.18 reaffirms that the maximum angle of incidence attained on the downstream pass is lower than that achieved on the upstream pass, which is a result of the expansion of the flow as it passes through the turbine.

In fact, for the 2-bladed turbine, it is this difference of 2.1 degrees in the maximum angle of attack, that is responsible for the different impact of dynamic stall on the upstream (adverse effect) and downstream passages (beneficial effect); see Figure 4.13a. As the maximum incidence is higher on the upstream than on the downstream passage of the turbine, the adverse effects of the force hysteresis are more significant for both the lift and the drag on the upstream pass, as illustrated in Figure 4.18. It is clear that on the upstream passage of the turbine, there is large scale vortex shedding resulting in a

large increase in sectional drag and an erratic lift behaviour with respect to incidence.

Note also that on the downstream passage of the turbine there are two local peaks in C_L , one as α increases and the other as α decreases. These peaks correspond to the local torque maxima at $\theta \approx 230^\circ$ and 300° .

Despite the superior performance on the downstream passage of the turbine the net effect of dynamic stall at $\sigma = 0.019$ & 0.029 and $\lambda = 3$ is negative due to the poor performance on the upstream side of the turbine.

Also, it is noted that dynamic stall has been observed on blades of wind as well as water cross-flow turbines before, for example see Brochier et al. (1986) and Laneville and Vittecoq (1986). The tests of Fujisawa and Shibuya (2001) have shown that the appearance of dynamic stall had positive effects on the power generation, whilst it was noted that stall vortices did create some problems, too, such as aeroelastic vibrations, which increase the risk of fatigue failure of the blades.

Figure 4.19 shows the sectional lift and drag coefficients for the 2- and 3-bladed turbine at a high tip speed ratio; $\lambda = 8$.

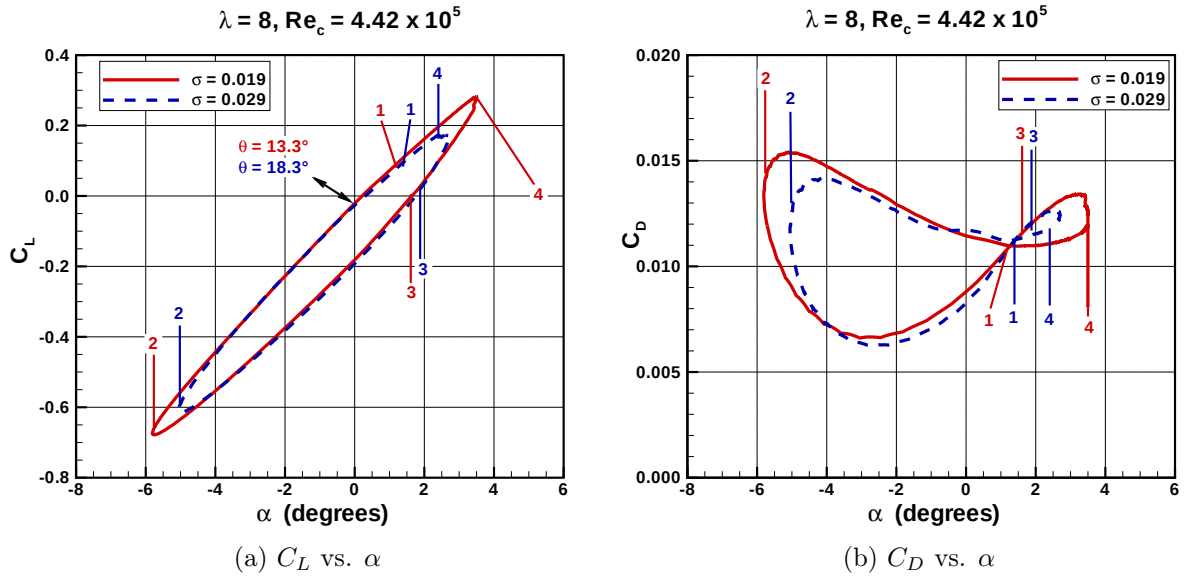


Figure 4.19: Blade coefficient histories for a blade of the 2- & 3-bladed turbine at $\lambda = 8$

Firstly, it is noted that both the lift and drag curves take a very similar shape for both solidities. In contrast to $\lambda = 3$, where blade stall results in a drop in lift and increase in drag, as the static/dynamic stall angle is passed, the loops observed in Figure 4.19 result in different C_L for the same α depending on whether α is increasing or decreasing. C_L for quadrants 2 & 4 is higher in comparison to quadrants 1 & 3. Likewise, there is a relative decrease in C_D for quadrants 2 & 4 in comparison to quadrants 1 & 3.

As discussed in Section 3.2.2.1, this loop is a result of shed vorticity and its effect on the incident flow field. In fact, the results shown in Figure 4.19 take a similar shape to the corresponding plots of the oscillating blade tests for $\alpha = 4^\circ \pm 4^\circ$ discussed in Section 3.2.2.1.

Also, as expected and as discussed for $\lambda = 3$, the blades of the lower solidity, σ , turbine are presented with a higher maximum angle of attack, $\alpha_{\max}$, due to the lower flow impedance relative to a higher solidity turbine, which results in higher α ; at $\lambda = 8$:

- $\alpha_{\max} = (-) 5.8^\circ$ for $\sigma = 0.019$,
- $\alpha_{\max} = (-) 5.0^\circ$ for $\sigma = 0.029$.

This also explains why $\alpha = 0^\circ$ occurs at an azimuth angle, θ , 5.0° larger for the 3-bladed turbine in comparison to the 2-bladed machine. The increase in solidity results in a reduction in streamwise flow velocity, which acts to reduce the angle of attack, so that $\alpha = 0^\circ$ is reached at a higher azimuth angle.

Similarly, it is interesting to note the difference in the azimuth position at which $\alpha = 0^\circ$ for the 2- and 3-blade tubing operating at $\lambda = 3$ & 8. An increase in λ results in a reduction in the streamwise flow velocity, and consequently, a reduction in α , so that $\alpha = 0^\circ$ occurs at a higher θ ; for instance, for the 2-bladed turbine:

- at $\lambda = 3$, $\alpha = 0^\circ$ is computed at $\theta = 5.3^\circ$,
- at $\lambda = 8$, $\alpha = 0^\circ$ is computed at $\theta = 13.3^\circ$.

As to the drag curves, it is observed that $\max C_D$ is lower for $\lambda = 8$ than for $\lambda = 3$, which may be explained by the lower angles of attack the blades of the turbine operate over at the higher λ . Since the static stall angle is not reached at $\lambda = 8$, the rapid increase in drag associated with blade stall and observed for $\lambda = 3$, does not occur at the higher end of the tip speed ratio range.

4.1.2.5 Turbine torque

Moreover, a comparison of the net (all blades) torque histories for the 2-, 3- and 4-bladed turbines has been conducted at $\lambda = 4$, see Figure 4.20. The 2-, 3- and 4-bladed turbines achieve mean torque coefficients, C_m , (per unit span of turbine) of 55.4, 69.0 and 74.0 respectively, which yield power coefficients, C_P , of 0.40, 0.50, 0.53.

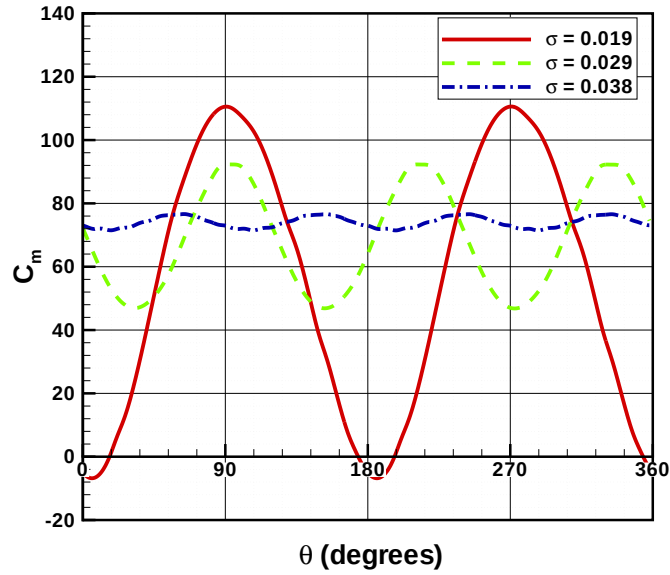


Figure 4.20: Comparison of turbine torque coefficient history at $\lambda = 4$ for $\sigma = 0.019$, 0.029 & 0.038

The key observation from Figure 4.20 is the considerably more uniform loading experienced by the 4-bladed turbine, which is a particularly favourable characteristic with regard to generator loading and fatigue issues. Although it should be noted that the

negative effects of increasing blade strut drag through increased number of blades has not been accounted for. Aside from the turbine's performance and its cost, which will partly depend on its solidity, turbine fatigue characteristics are important in identifying optimal turbine designs.

4.2 Reynolds number = $O(10^6)$

In Chapter 3 differences between the simulations conducted at $Re_c = O(10^5)$ & (10^6) were noted, both in terms of the performance of the NACA 0015 blade section in the static blade tests as well as the required mesh resolution to ensure grid convergence.

Moreover, it is of interest to further the understanding of the differences in performance of a cross-flow turbine at $Re_c = O(10^5)$ & (10^6) , so that results obtained from physical experiments carried out at a high laboratory Reynolds number may be interpreted appropriately with regard to the performance of the same turbine operating at a field-test or even full-scale Re . For instance, the case of the turbine with $\sigma = 0.019$ operating at $\lambda = 3$, for which the occurrence of dynamic stall was observed at $Re_c = 4.42 \times 10^5$, was considered of particular interest for higher Re simulations to examine how the performance of a turbine may change as Re is varied.

Hence, it was determined to repeat the above solidity investigation at a higher blade Reynolds number, $Re_c = O(10^6)$. The only parameters that were changed in comparison to the simulations presented in Section 4.1 were the fluid's dynamic viscosity, which was adjusted so that $Re_c = 2 \times 10^6$, as well the grid employed for the discrete circular domains around each blade.

From the tests conducted in Section 3.2, it was concluded that Mesh VI_a was best suited for simulations at $Re_c = 2 \times 10^6$; Mesh 2, which was used for the domain around the blades for the lower Re simulations, was hence replaced with Mesh VI_a. For Mesh VI_a, the first grid spacing from the surface in the normal direction was $5 \times 10^{-5} c$, which resulted in $y^+ = 1.6$ at $\alpha = 2.5^\circ$. With a streamwise mesh resolution of 160 cells per blade side this resulted in 50,000 cells for the circular domain around each blade and in a total number of elements in the entire turbine domain of 138,800 for the 2-bladed, 190,900 for the 3-bladed and 244,600 for the 4-bladed turbine configurations.

4.2.1 Solution convergence

The same procedure presented in Section 4.1.1 was followed to determine solution convergence for the simulations at the higher Re . To enable a direct comparison with the lower Re simulations, one example, including the analysis of a turbine's C_P trace, as well as wake velocity plots has been provided below for $\sigma = 0.019$ and $\lambda = 8$ at both Re , see Figure 4.21 & 4.22:

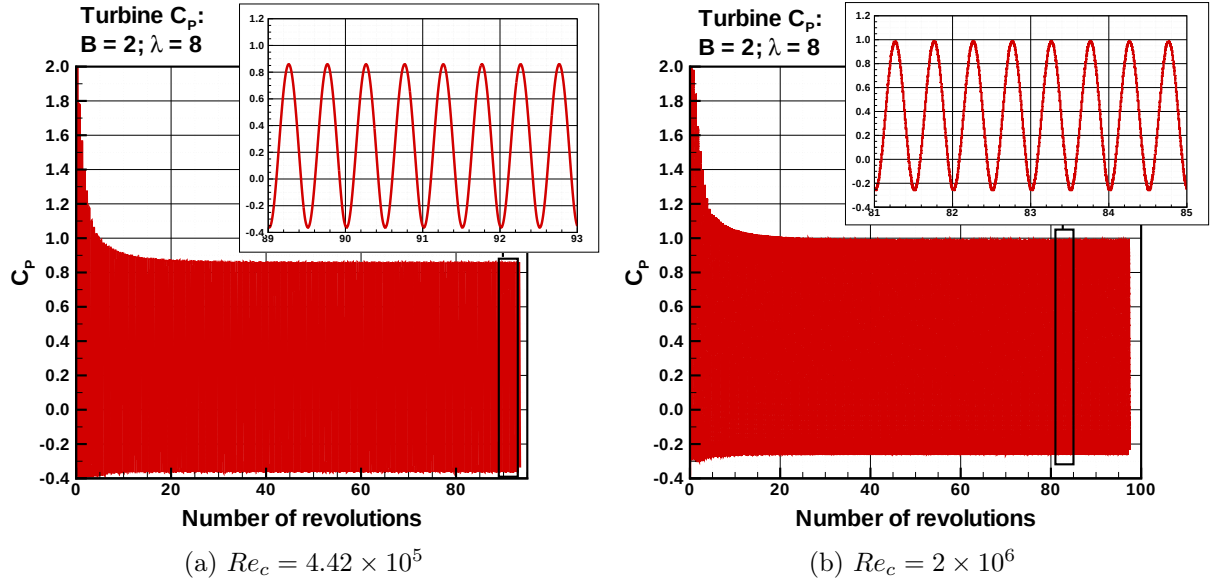
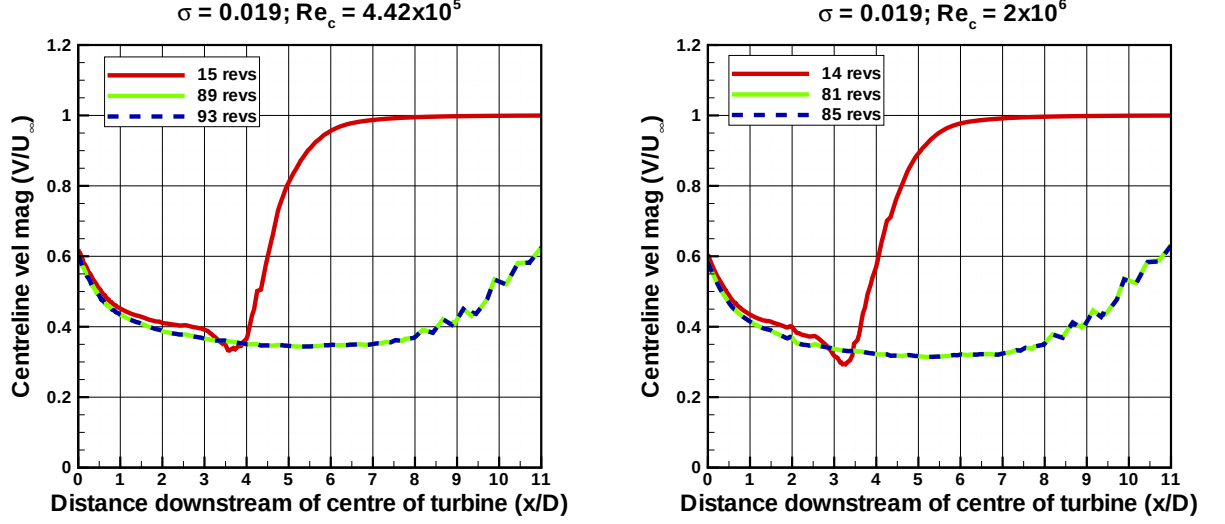


Figure 4.21: Convergence of turbine power histories for $B = 2$, $\sigma = 0.019$ and $\lambda = 8$

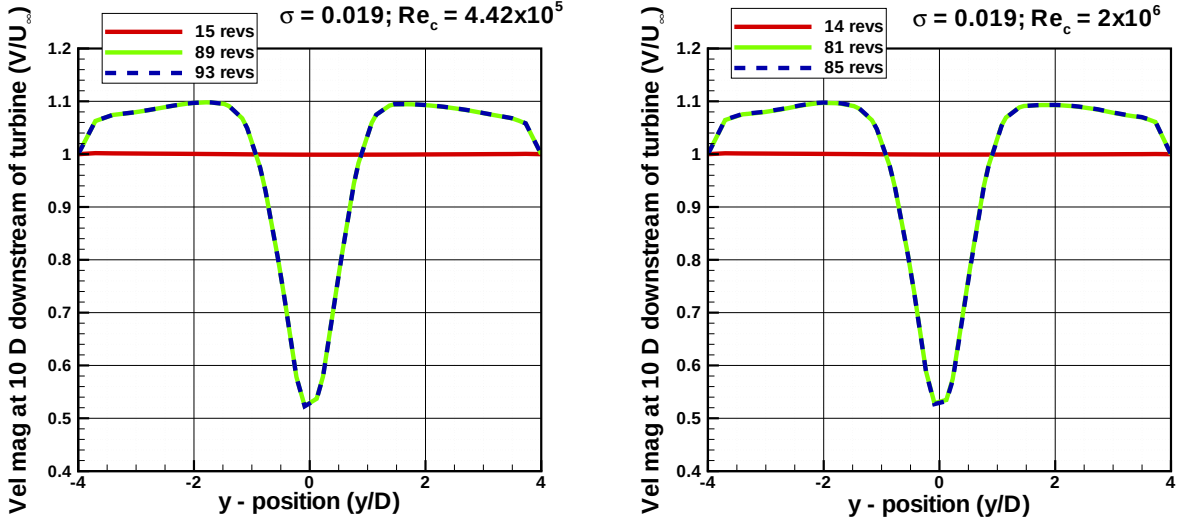
The difference in magnitude in C_P is discussed in detail below. As to the form of the curves, there are very little differences arising from the difference in Re . In both cases, C_P is considered statistically converged after 30 cycles, but the wake converged after fewer revolutions for the larger Re , see Figure 4.22, which shows both the centreline velocity magnitude as well as a cross-stream traverse of the velocity magnitude at 10 D downstream of the turbine for both Re and $\sigma = 0.019$, $\lambda = 8$.

The shapes of the curves are as expected and similar to the corresponding cases discussed previously. As for C_P , the differences in velocity appear small for the different Re simulations, but the wake of the higher Re case converges more rapidly, after 81

cycles compared to 89 in the lower Re case. This we attribute to increased wake mixing at higher Re and therefore a more rapid route to the steady-state solution.



(a) Centreline velocity magnitude at $Re_c = 4.42 \times 10^5$ (b) Centreline velocity magnitude at $Re_c = 2 \times 10^6$



(c) Velocity magnitude for a cross-stream traverse at 10 D downstream of turbine at $Re_c = 4.42 \times 10^5$ (d) Velocity magnitude for a cross-stream traverse at 10 D downstream of turbine at $Re_c = 2 \times 10^6$

Figure 4.22: Wake velocity profiles for $B = 2$, $\sigma = 0.019$ and $\lambda = 8$

The time required to convergence in C_P is broadly similar for both Re , as shown in Figure 4.23, which shows the number of revolutions required until C_P was unchanged for 10 consecutive revolutions, for a range of tip speed ratios simulated for the three

different turbine solidities at $Re_c = 4.42 \times 10^5$ & 2×10^6 . Similar observations may be made at $Re_c = 2 \times 10^6$ to those identified at lower Re ; the number of revolutions required until convergence increases, as turbine solidity increases. Moreover, it may be noted that the larger λ , the more revolutions required for solution convergence.

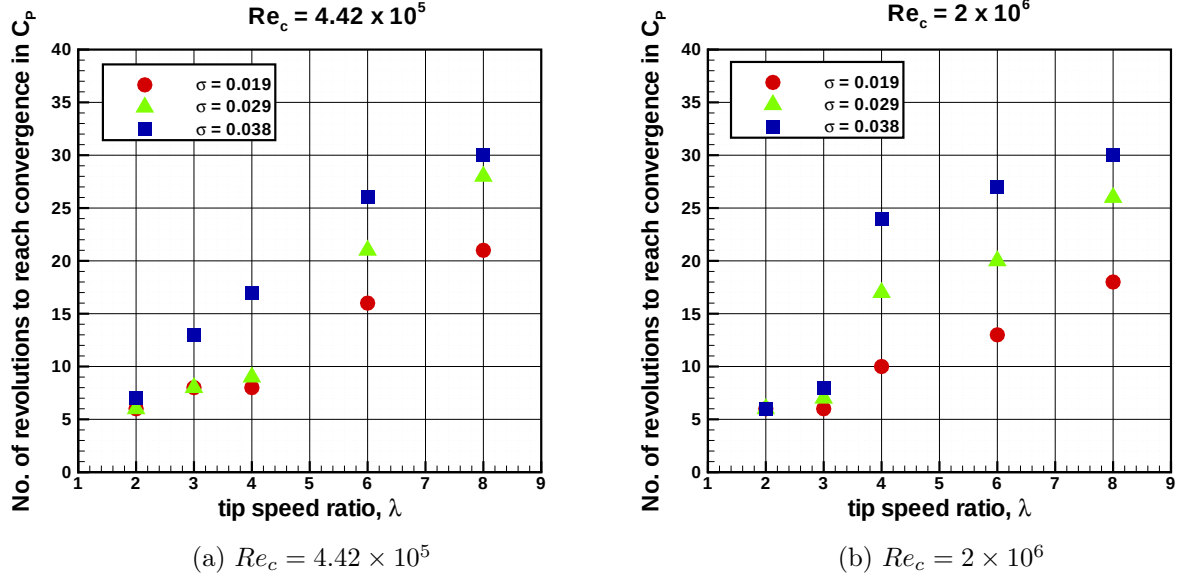


Figure 4.23: Number of revolutions for convergence in C_p

4.2.2 Results

Figure 4.24 shows the power and thrust curves for the turbines with solidities, $\sigma = 0.019$, 0.029 and 0.038 , operating at $Re_c = 2 \times 10^6$. The shapes of the power and thrust curves as well as the changes observed when varying σ are very similar to the results obtained for $Re_c = 4.42 \times 10^5$. In order to enable a direct comparison to the corresponding simulations at $Re_c = 4.42 \times 10^5$, the power curves for the same turbine solidity, but different Re have been plotted on individual plots in Figure 4.25.

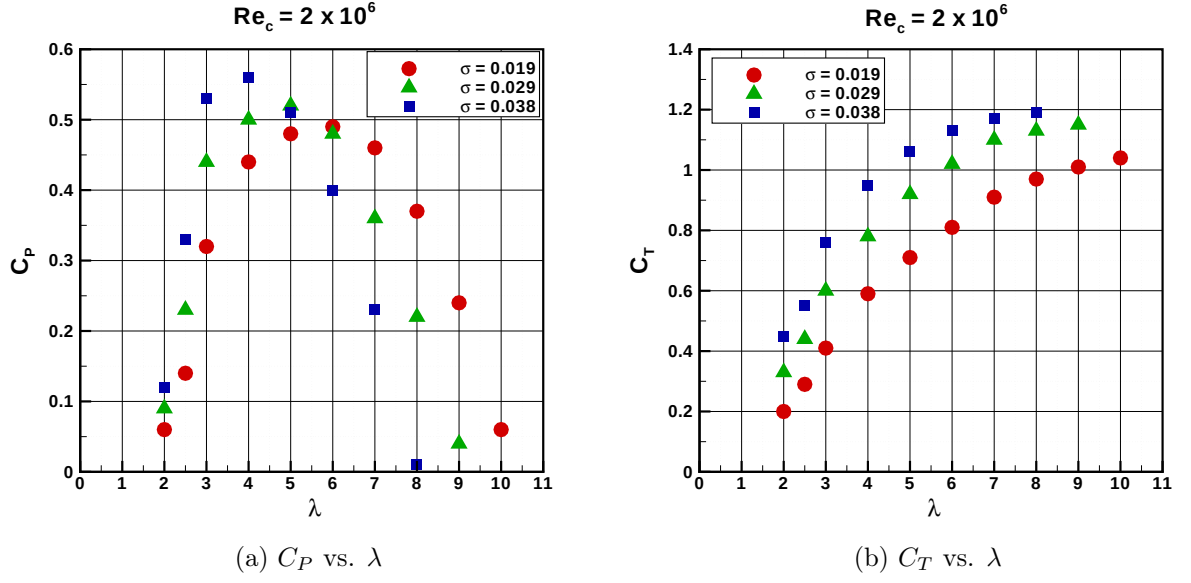


Figure 4.24: Power and thrust coefficient variation for varying turbine solidity, σ , at $Re_c = 2 \times 10^6$

Firstly, for all three σ and at most λ , an increase in Re_c from 4.42×10^5 to 2×10^6 results in an increase in C_P & C_T . For instance, for $\sigma = 0.019$, max C_P increases from 0.43 to 0.49, i.e. by 14.0%, and by 4.0% for $\sigma = 0.029$ and 5.7% for $\sigma = 0.038$. Moreover, a widening and shift of curve to a higher range of λ is observed at higher Re . The relative increase and width and shift of the curve is largest for the 2-bladed turbine, see Figure 4.25a.

The principal driver for the increase in C_P across the tip speed ratio range is that the lift-to-drag ratio is higher at high Re . Below, the effect of increasing Re on low λ and then on high λ is discussed in detail.

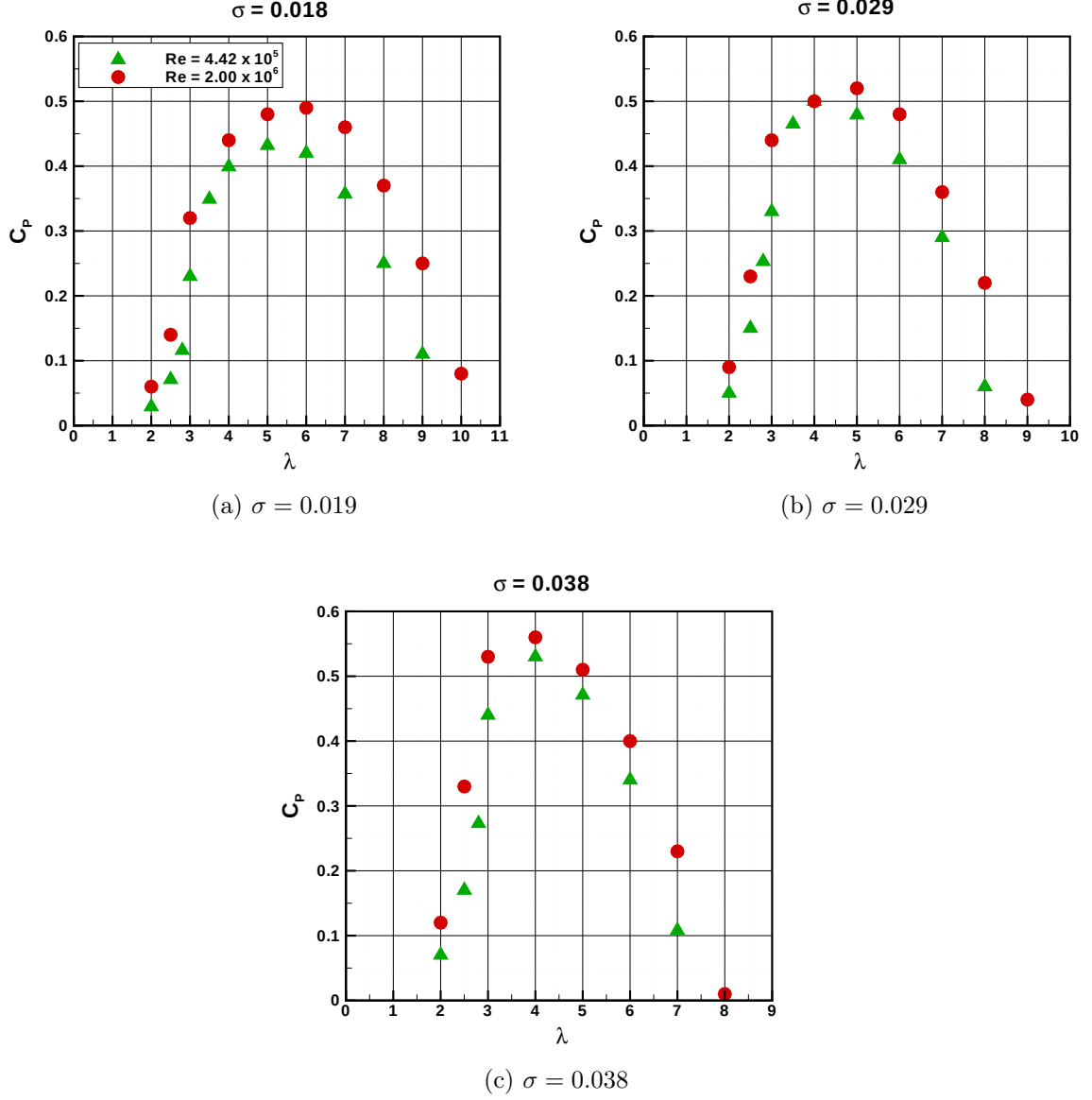


Figure 4.25: Comparison of power coefficients for different solidities at $Re_c = 4.42 \times 10^5$ & $Re_c = 2 \times 10^6$

The increase in C_p for higher Re at low λ , where the power take-off is limited by blade stall, may be explained by the improved performance at higher Re of a blade operating at high angles of attack, as discussed in Section 3.2. To examine this further, we study the lift and drag curves for $\sigma = 0.019$ & 0.029 at $\lambda = 3$ for both Re , see Figure 4.26 & 4.27. In the manner plotted, positive incidence refers to the downstream pass of the blade, whilst negative incidence refers to the upstream blade pass.

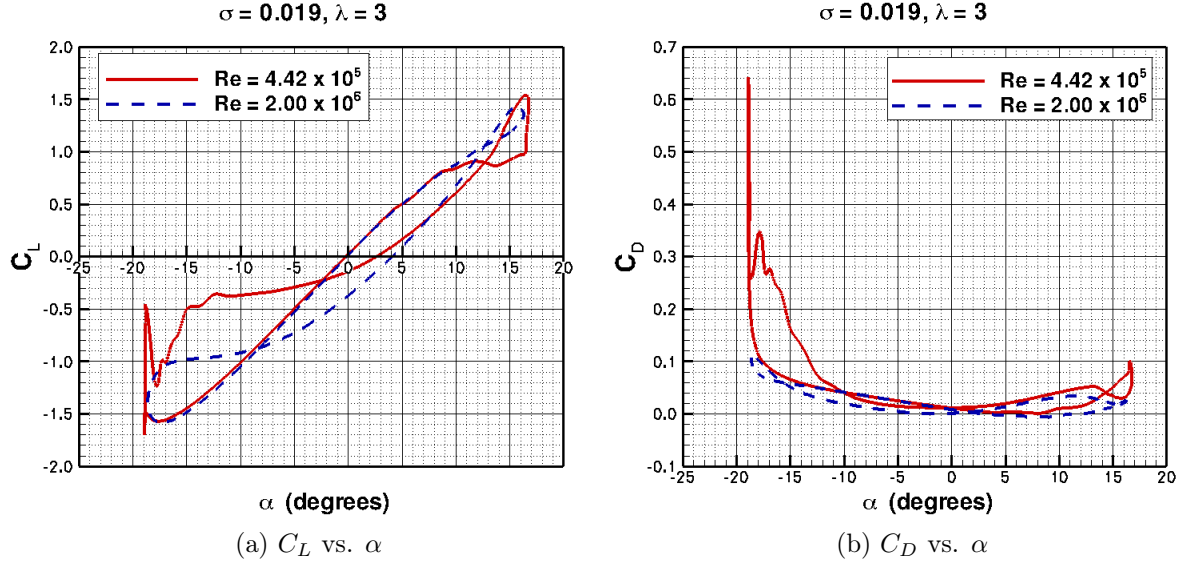


Figure 4.26: Blade coefficient histories for a blade of the 2-bladed turbine at $\lambda = 3$ operating at $Re_c = 4.42 \times 10^5$ & $Re_c = 2 \times 10^6$

At $\lambda = 3$ and $\sigma = 0.019$, C_P was computed to be 39.1% larger for the higher Re case. Figure 4.26 shows that the negative effects associated with blade stall, the drop in lift and increase in drag, are significantly less severe for the higher Re simulations, although the maximum angle of attack, α , is very similar at both Re on both the upstream and downstream passages. Moreover, it is apparent that reattachment takes place at a significantly lower α for the lower Re simulation, resulting in a wider (lift) hysteresis loop. Similarly, the blades of the turbine operating at the higher Re outperform the blades of the turbine operating at the lower Re on the downstream passage.

Similar observations can be drawn from the case of $\sigma = 0.029$ at $\lambda = 3$; see Figure 4.27.

Moreover, it is noted that the C_m trace of a single blade operating at a low λ is smoother and more symmetric for the higher Re simulations, for example see Figure 4.28a. This is because the erratic oscillations of the blade forces initiated by dynamic stall observed for the low Re simulations for $\sigma = 0.019$ & 0.029 at $\lambda = 3$, as discussed in Section 4.1.2.4, do not occur at $Re_c = 2 \times 10^6$. In addition to the increase in power, this is particularly

beneficial with regard to potential fatigue issues.

It is of interest to note that for the higher solidity analysed, $\sigma = 0.029$, see Figure 4.27, $\max \alpha$ on the downstream passage is lower for the higher Re simulations. This is due to the increase in flow impedance presented by the turbine operating at the higher Re ; C_T is 11.1% larger for $\sigma = 0.029$ at $\lambda = 3$. This is confirmed by comparing cross-stream velocity magnitudes at $1 D$ downstream of the centre of the turbine, see Figure 4.28b, which shows that the core flow velocity is lower for the higher Re case.

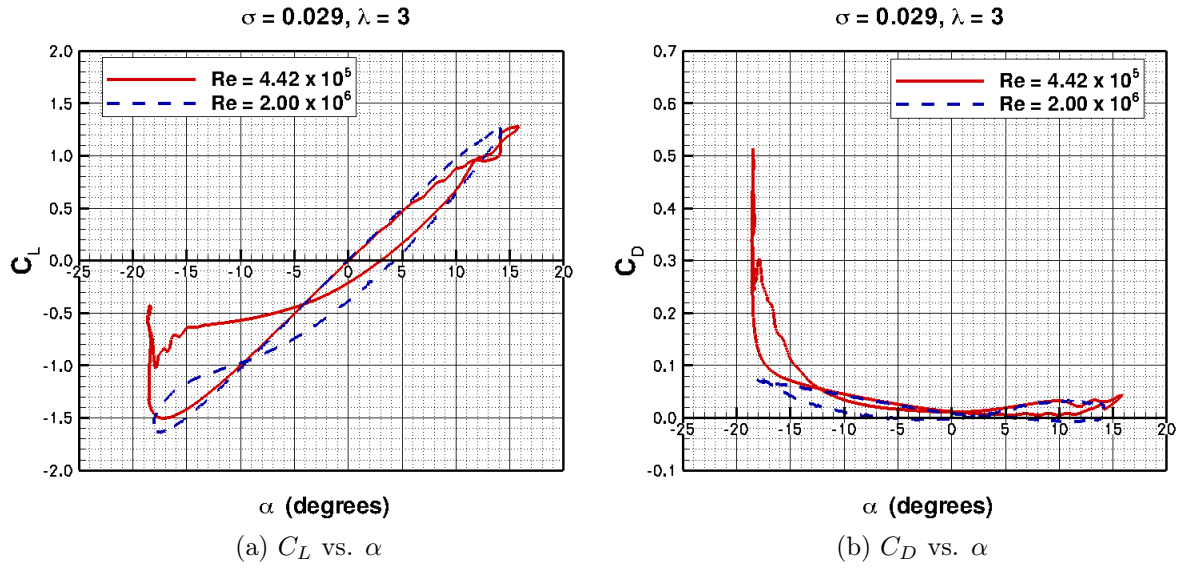


Figure 4.27: Blade coefficient histories for a blade of the 3-bladed turbine at $\lambda = 3$ operating at $Re_c = 4.42 \times 10^5$ & $Re_c = 2 \times 10^6$

Also, it is apparent that the velocity profile of the higher Re computations is smoother, which is due in part to the less erratic vortex shedding, as observed in the lift, drag and C_m plots, and in part due to viscous effects at high Re acting to increase mixing through the flow field.

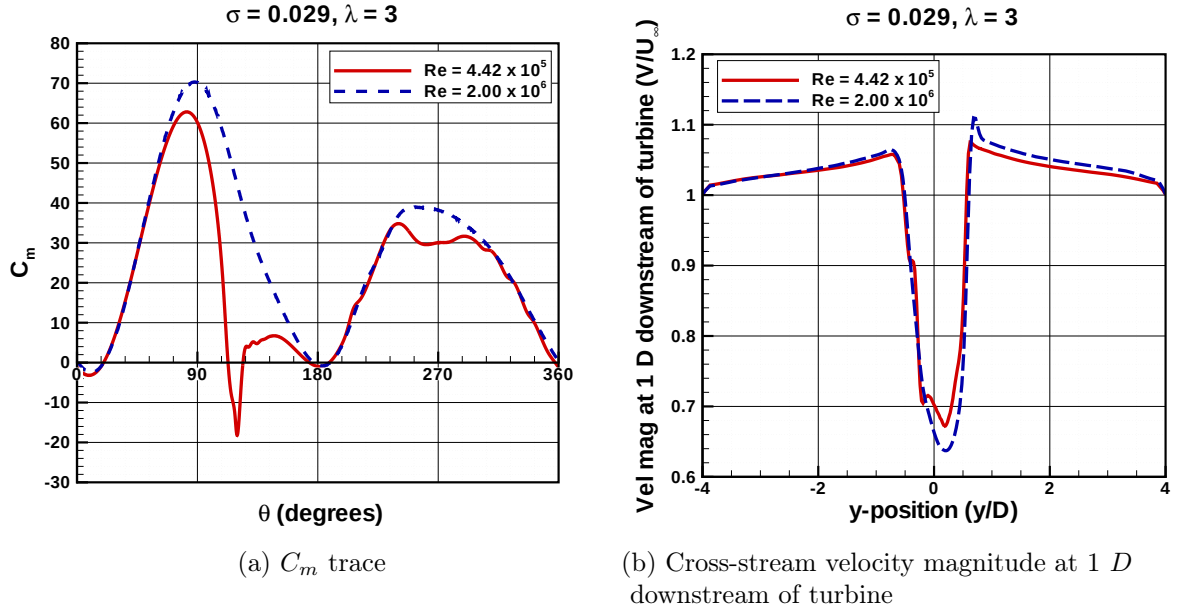


Figure 4.28: Various comparisons for $\sigma = 0.029$ and $\lambda = 3$ at $Re_c = 4.42 \times 10^5$ & $Re_c = 2 \times 10^6$

Moreover, the increase in flow impedance observed for the higher Re simulations leads to a shift of the upstream peak of the C_m trace to a higher azimuth angle, θ ;

- $\theta = 82.0^\circ$ at max C_m for the lower Re simulation,
- $\theta = 89.4^\circ$ at max C_m for the higher Re simulation.

A reduction of the streamwise flow velocity acts to reduce the angle of attack, so that the static stall angle, α_{ss} , or equivalently, the eventual dynamic stall angle, is reached at a later azimuth.

Furthermore, the effect of increasing Re on the performance of a cross-flow turbine at higher tip speed ratios was examined. The power take-off also increases with an increase in Re from 4.42×10^5 to 2×10^6 at high λ , where the limiting factor is not blade stall, but low angles of attack.

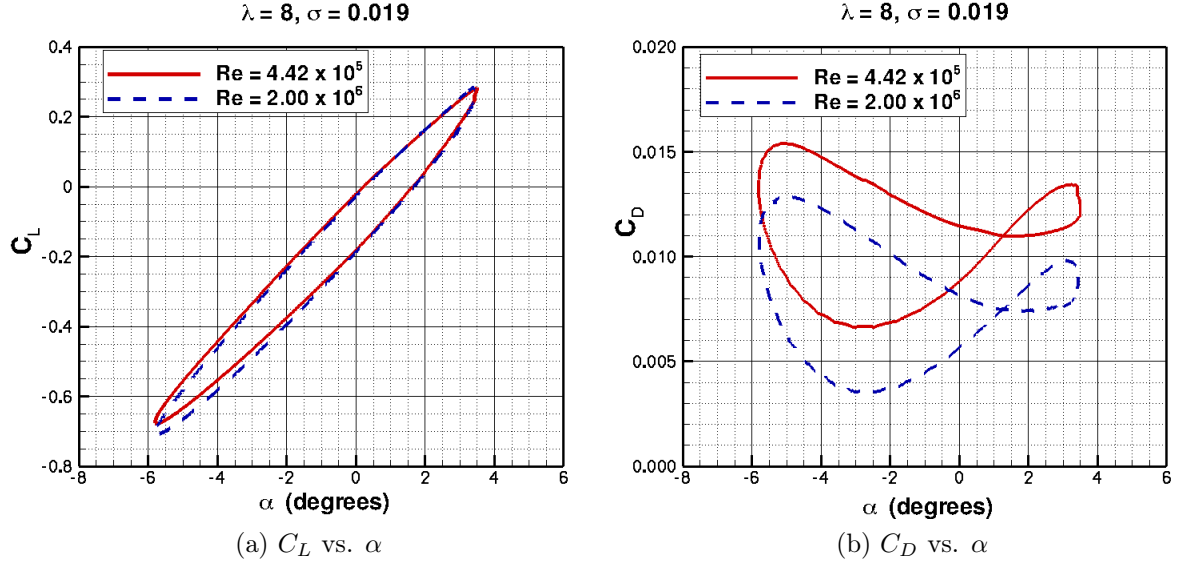


Figure 4.29: Blade coefficient histories for a blade of the 2-bladed turbine at $\lambda = 8$ operating at $Re_c = 4.42 \times 10^5$ & $Re_c = 2 \times 10^6$

From Figure 4.29 it is apparent that the lift curves for $\sigma = 0.019$ and $\lambda = 8$ are similar for both Re . The gradient of the C_L vs. α plot is slightly higher for the higher Re simulation, resulting in superior lift on the upstream passage.

When comparing the drag curves, an offset of about 0.003 is observed. It is this offset in C_D , that explains the offset observed when comparing the C_m traces in Figure 4.30a. The comparison of C_m traces at the high end of the tip speed ratio range renders two curves, which overlap at no position throughout the cycle. The blade operating at the higher Re generates more torque at all azimuth angles. The largest difference occurs at the peak of the C_m traces, at $\theta = 97^\circ$, which is in part due to the higher lift generated at max α on the upstream passage. However, for the majority of the cycle, the lift generated by the blades operating at different Re is very similar, see Figure 4.30b.

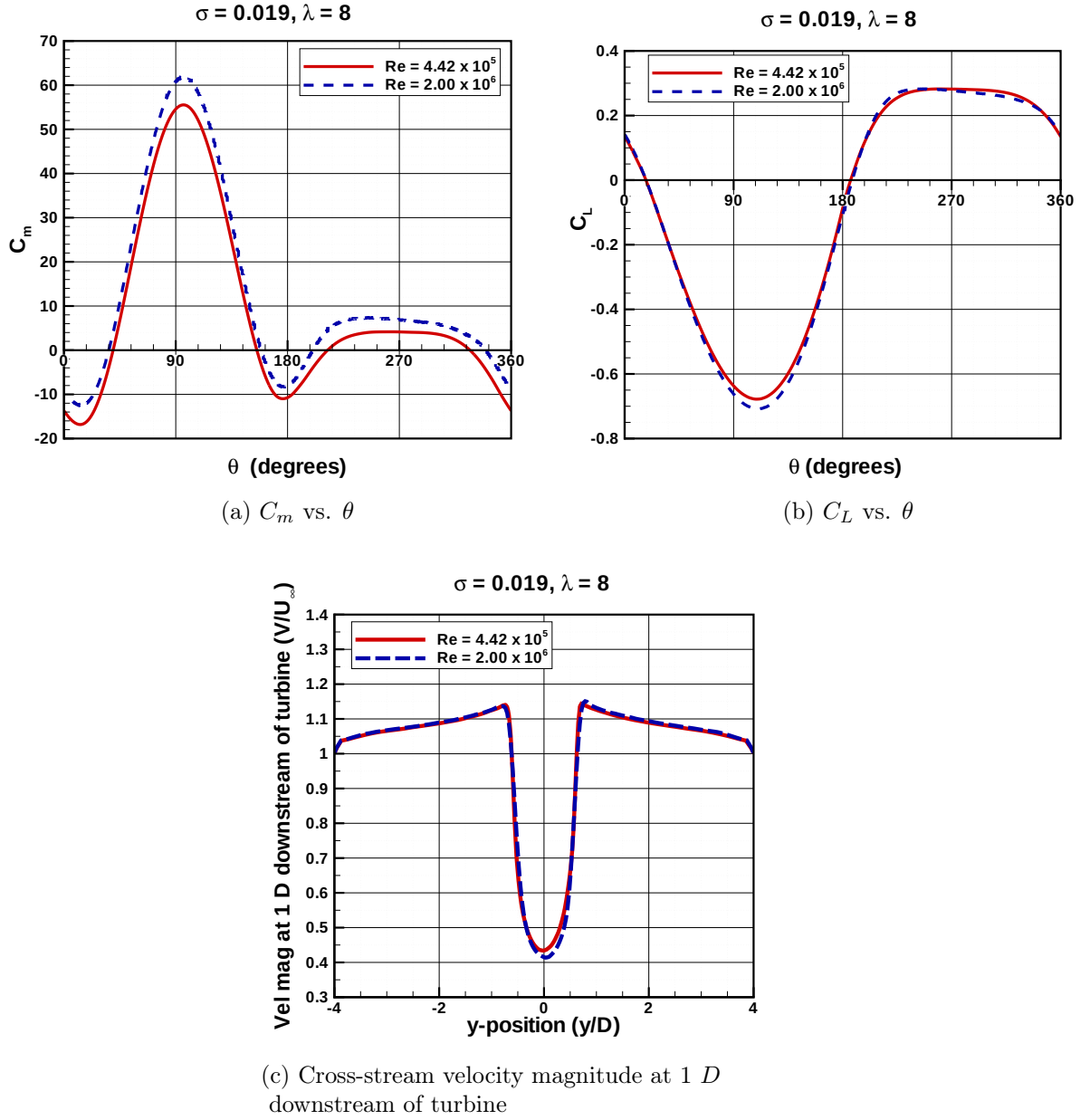


Figure 4.30: Various comparisons for $\sigma = 0.019$ and $\lambda = 8$ at $Re_c = 4.42 \times 10^5$ & $Re_c = 2 \times 10^6$

It is of interest to note that the shift in the peak of the C_m traces of the different Re simulations is smaller for $\sigma = 0.019$ and $\lambda = 8$ than for $\sigma = 0.029$ and $\lambda = 3$. This is because the difference in the streamwise flow velocity for the different Re computations is smaller, too, for $\sigma = 0.019$ and $\lambda = 8$; as highlighted in Figure 4.30c, at 1 D downstream of the turbine the difference in the minimum velocity magnitude between

the different Re simulations, $\Delta V_{\min}/U_{\infty} = 0.021$ for $\sigma = 0.019$ and $\lambda = 8$, whilst $\Delta V_{\min}/U_{\infty} = 0.036$ for $\sigma = 0.029$ and $\lambda = 3$. The smaller difference in $\Delta V_{\min}/U_{\infty}$ for the different Re simulations for $\sigma = 0.019$ and $\lambda = 8$ is due to a smaller difference in C_T indicating a smaller difference in flow impedance; for $\sigma = 0.019$ and $\lambda = 8$, the difference in the thrust coefficients, $\Delta C_T = 0.02$, whilst $\Delta C_T = 0.10$ for $\sigma = 0.029$ and $\lambda = 3$. Because the flow impedance and thus the streamwise flow velocity is very similar for the different Re simulations for $\sigma = 0.019$ and $\lambda = 8$, the azimuth position, θ , at which the maximum angle of attack is reached and thus maximum torque is generated is similar; $\max C_m$ occurs at $\theta = 97.2^\circ$ for the lower Re and at $\theta = 97.5^\circ$ for the higher Re computation.

Above it was concluded that lift generated by the individual blades of the turbine with $\sigma = 0.019$ operating at $\lambda = 8$ is very similar for both Re simulations. This suggests that the higher torque observed for the higher Re simulations at high λ , see Figure 4.30a, is due to the lower drag observed for the higher Re simulations. Further comparisons of the lift, drag and C_m traces at high λ for different Re have affirmed this; for example, see Figure 4.31, which shows the lift and drag curves for $\sigma = 0.029$ and $\lambda = 8$ at both Re examined in the present study. As above, the lift curves are similar, with a slightly higher gradient for the higher Re simulation, while an offset in C_D of about 0.003 is observed.

The decrease in C_D for higher Re observed for the present numerical simulations concurs with the corresponding results from physical experiments; for instance, when examining the data for C_D from static blade tests conducted for a NACA 0015 blade section at varying Re in Sheldahl and Klimas (1981), see Figure 3.8b, it is apparent that drag decreases with increasing Re . Moreover, in Abbott and von Doenhoff (1959) it is discussed that C_D generally decreases with increasing Re up to $Re_c = 2 \times 10^7$.

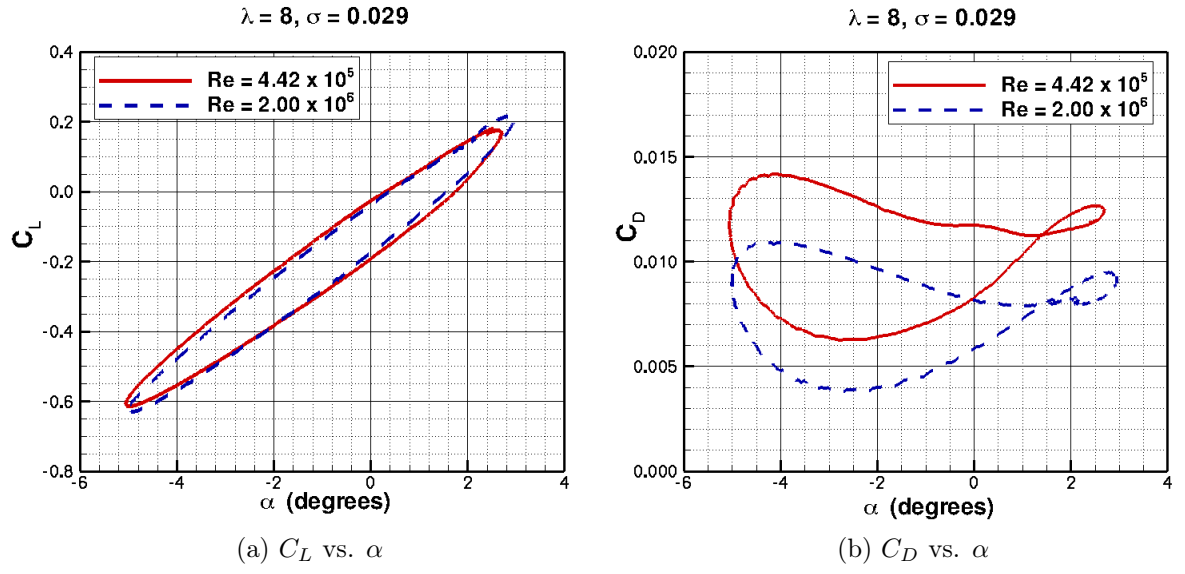


Figure 4.31: Blade coefficient histories for a blade of the 3-bladed turbine at $\lambda = 8$ operating at $Re_c = 4.42 \times 10^5$ & $Re_c = 2 \times 10^6$

4.3 Chapter conclusions

In this chapter we have explored the influence of turbine solidity on the hydrodynamics and turbine performance of cross-flow turbines. The investigation was conducted for two Reynolds numbers, $Re_c = O(10^5)$ & $O(10^6)$, to reflect laboratory and field scales. Further the investigation was for a modest blockage ratio of 12.5%.

Increasing the number of blades from two to four led to an increase in the maximum power coefficient from 0.43 to 0.53 for the lower Re and from 0.49 to 0.56 for the higher Re computations. Moreover, the power curve was found to shift to a lower range of tip speed ratios when increasing turbine solidity.

Also, it was observed that dynamic stall occurred at the lowest tip speed ratios for the lower Re simulations on both the upstream and downstream blade passes. However, the net effect of dynamic stall on turbine performance was shown to be negative for the turbine configuration investigated.

With regard to the Re comparison, it was found that, in addition to a slight increase in power, an increase in Re resulted in a widening and shift of the power curve to a higher range of tip speed ratios.

Chapter 5

Turbines in confined flow

5.1 Flow confinement study

This section presents an investigation of flow confinement effects on the hydrodynamics of a generic tidal cross-flow turbine. The dependency of the flow-field on the location of the numerical model's cross-stream boundaries is examined and, in particular, the effects on turbine performance are investigated.

The turbine design used for the present computations was based on the turbine configuration tested experimentally by McAdam et al. (2010):

Number of blades	B	3
Chord length	c	0.0654 m
Rotor radius	R	0.25 m
Turbine solidity	σ	0.125

Table 5.1: Parameters of a turbine tested by McAdam et al. (2010)

where turbine solidity, σ , is defined as:

$$\sigma = \frac{cB}{2\pi R} \quad (5.1.1)$$

where B , c and R are the number of blades, the blade chord length and the turbine radius respectively.

However, two parameters were changed for the simulations:

- blade section and
- mean blade Reynolds number, Re_c .

McAdam et al. (2010) used a Clarkysimm hydrofoil section with a thickness of 18.3%, for further details on this hydrofoil see Consul (2008), whilst a NACA 0015 was used for the numerical tests, following on from the validation discussed in Chapter 3 on the NACA 0015 section. Also, the Reynolds number was matched to that of the validation stage outlined in Section 3.2, which had been chosen to reflect a realistic tidal turbine installation; for the present turbine simulations, $4 \times 10^5 < Re_c < 3.6 \times 10^6$, depending on tip speed ratio, λ , where $\rho = 1030 \text{ kg/m}^3$ and $\mu = 0.002575 \text{ kg/m} \cdot \text{s}$ and where:

$$\lambda = \frac{\omega R}{U_\infty} \quad (5.1.2)$$

where ω is the turbine angular velocity and U_∞ the free-stream velocity.

It was decided to match the blade section and Re_c of the validation tests and the present turbine simulations, so that the same mesh employed for the static and dynamic blade tests in Section 3.2 could be used for the full turbine simulations.

The computational domain, a two-dimensional slice orthogonal to the turbine's axis of rotation, is made up of three sub-domains, as illustrated in Figure 5.1a:

1. a far-field domain, shown in orange,
2. a turbine domain consisting of a circular rotating mesh, shown in green, and
3. discrete circular domains around each blade, shown in red.

Because the computational domain is two-dimensional, the turbine blades are implicitly assumed to be infinitely long. A consequence of this assumption is that drag due to end supports (plates or struts) is assumed negligible.

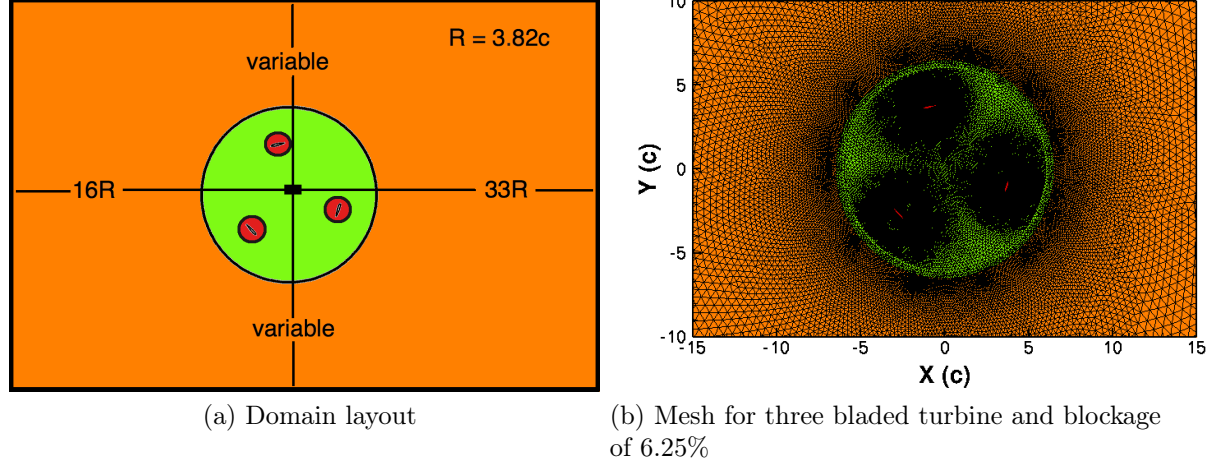


Figure 5.1: Computational domain for the present flow confinement investigation

For the discrete circular domain around each blade Mesh VI_a from Section 3.2 was used, a hybrid mesh comprising a very fine structured quadrilateral element grid for the near-wall region, a fine structured grid for the region within $1.5 c$ of the hydrofoil and an unstructured triangular grid for the remainder of the domain, see Figure 5.1b. Each hydrofoil surface was discretised with a total of 160 cells per blade side. The first grid spacing from the surface in the wall normal direction was $5 \times 10^{-5} c$, which resulted in $0.2 \leq y^+ \leq 3$ depending on blade incidence.

The turbine is of radius $R = 3.82 c$. The domain extends $16 R$ upstream and $33 R$ downstream of the centre of the turbine. For the present investigation four different set-ups were tested, as detailed in Table 5.2, resulting in blockages of 6.25%, 12.5%, 25% and 50%, where blockage, b , is the ratio of the diameter of the turbine, $2 R$, to the width of the domain, h .

Test	Width of domain, h	Blockage, b	No. of elements
1	32 R	6.25%	210,800
2	16 R	12.5%	206,000
3	8 R	25%	204,900
4	4 R	50%	187,800

Table 5.2: Details of the test cases for the flow confinement investigation

For the widest mesh, the domain extends 16 R laterally to either side of the turbine, 8 R laterally to either side of the turbine for $b = 12.5\%$ and 4 R laterally to either side of the turbine for $b = 25\%$. In tests 1 - 3 the turbine is placed at mid-depth, but in test 4 the turbine is placed, so that the domain extends 2.3 R laterally in the positive y -direction and 1.7 R laterally in the negative y -direction; the last configuration is identical to that tested by McAdam et al. (2010).

Using Mesh VI_a resulted in a mesh size of 68,400 for the circular domains containing each blade, see Figure 5.1, which resulted in meshes of around 200,000 elements in each case, as shown in Table 5.2.

The following boundary conditions were employed in the present simulations:

1. no slip condition on the blade surfaces,
2. unperturbed streamwise flow conditions across the upstream (inflow) boundary,
i.e. $U_x = 1 \text{ m/s}$ and $U_y = 0 \text{ m/s}$,
3. symmetry boundary conditions for the cross-stream boundaries and
4. a constant pressure condition along the downstream (outflow) boundary, i.e.
 $P = 0 \text{ Pa}$.

where U_x and U_y are the streamwise and cross-stream velocities respectively.

To simulate the rotation of the rotor, the circular turbine mesh with embedded blades is prescribed to move relative to the outer inertially fixed domain. For the numerical

discretisation, the PISO scheme was employed for pressure-velocity coupling, second order upwind for the momentum equations as well as for the turbulence modelling equations.

5.1.1 Solution convergence

The number of revolutions required to attain a statistically converged solution varied significantly for the differently blocked cases and was also found to depend on tip speed ratio, λ . In order to determine solution convergence, the blade forces and the development of the wake were examined cycle-to-cycle. Firstly, the torque coefficient, C_m , of a single blade, and the power coefficient, C_P , of the full turbine are compared, see Figure 5.2, 5.4 & 5.6, and secondly, the centreline wake velocity magnitude and the velocity magnitude at cross-stream wake traverses at 1 diameter, D , 5 D and 10 D downstream of the turbine are examined, see Figure 5.3, 5.5 & 5.7. For blockages of 50%, 12.5% and 6.25%, an example of solution convergence is given corresponding to the tip speed ratio of maximum power take-off, λ_{opt} .

A simulation was considered statistically converged, when, for 10 consecutive revolutions, turbine C_P did not change to the second significant figure and the various wake velocity plots remained unaltered for 5 consecutive revolutions.

As illustrated in Figure 5.2 - 5.7, the least number of revolutions to attain solution convergence were required for the 50% blocked simulations. For $b = 50\%$ and $\lambda = 3$, the solution was considered statistically converged after 21 revolutions, see Figure 5.2 & 5.3.

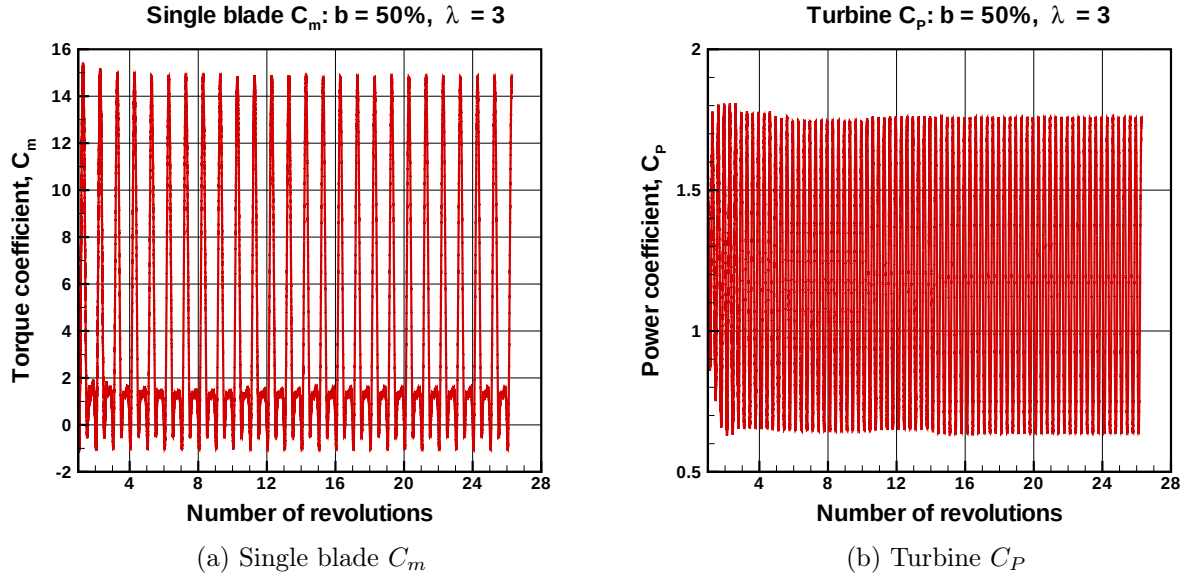
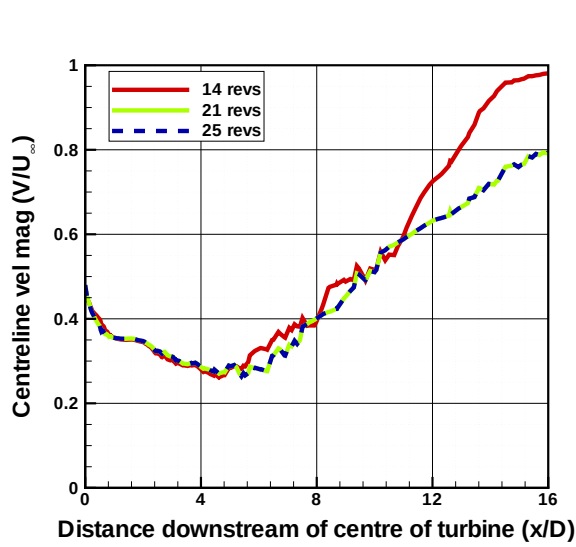


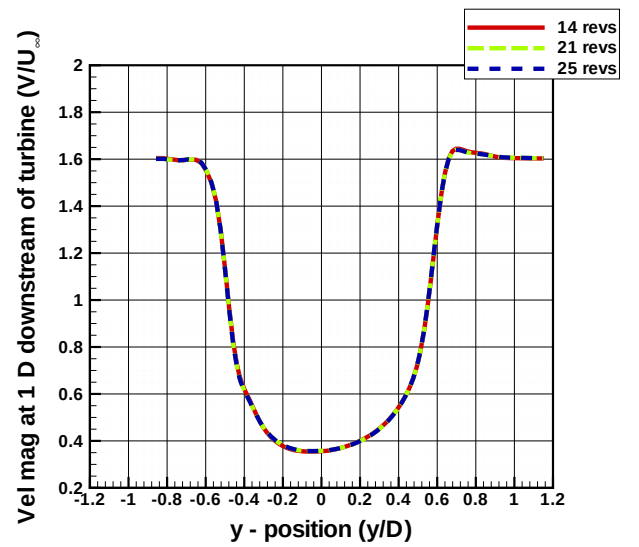
Figure 5.2: Convergence of blade torque and turbine power histories for $b = 50\%$ and $\lambda = 3$

With regard to the blade moments, Figure 5.2a & 5.2b show that the solution is statistically converged after 14 revolutions. However, 7 further revolutions were required to reach convergence in the wake flow-field.

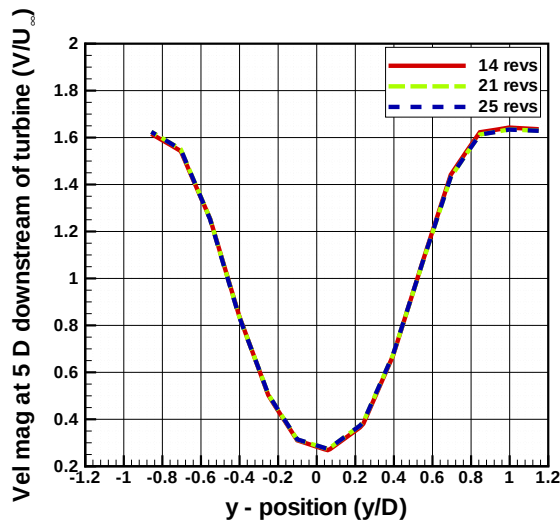
Figure 5.3b & 5.3c illustrate that the velocity magnitude at the cross-stream traverses at $1 D$ and $5 D$ downstream of the centre of the turbine did not change after 14 revolutions. But, as apparent in Figure 5.3a & 5.3d, the centreline velocity magnitude as well as the velocity magnitude at the cross-stream traverse at $10 D$ downstream of the turbine were still varying from cycle-to-cycle and did not reach convergence until revolution 21. Due to wake convection convergence takes longer, as the flow field is examined further downstream.



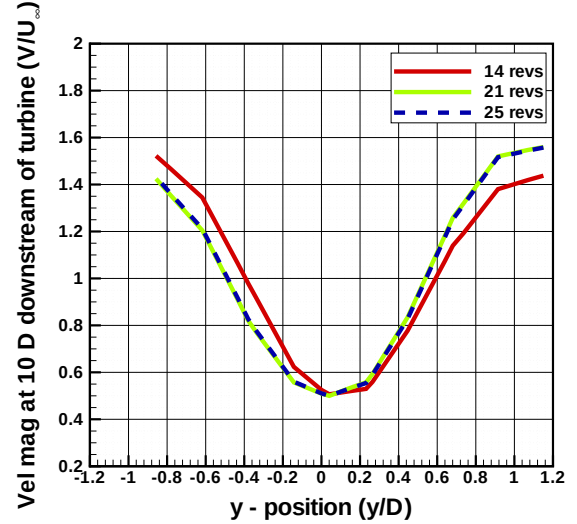
(a) Centreline velocity magnitude



(b) Velocity magnitude for a cross-stream traverse at 1 D downstream of turbine



(c) Velocity magnitude for a cross-stream traverse at 5 D downstream of turbine



(d) Velocity magnitude for a cross-stream traverse at 10 D downstream of turbine

Figure 5.3: Wake velocity profiles for $b = 50\%$ and $\lambda = 3$

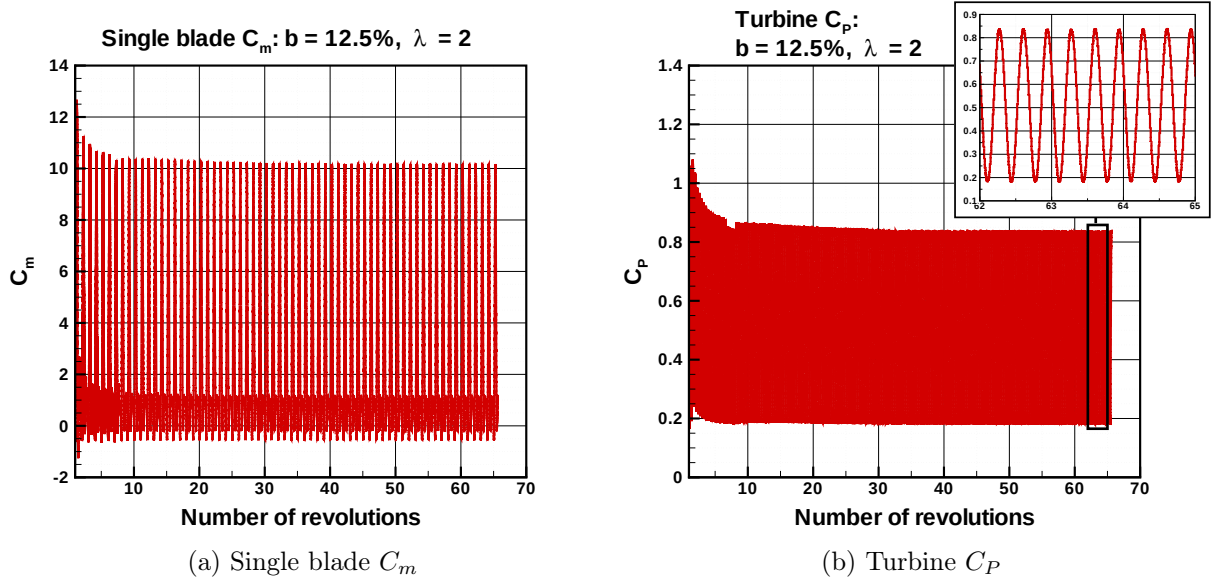
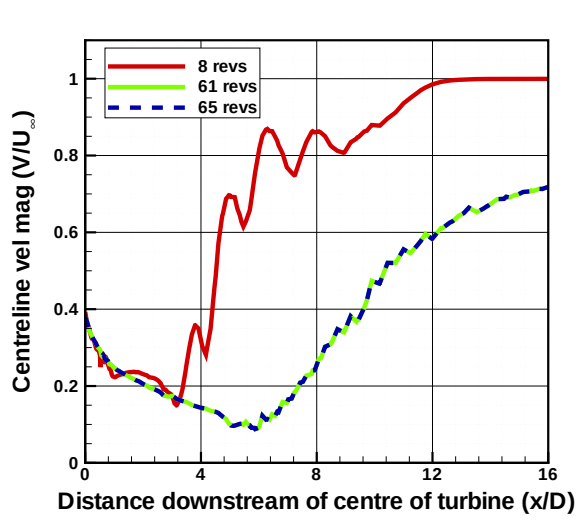


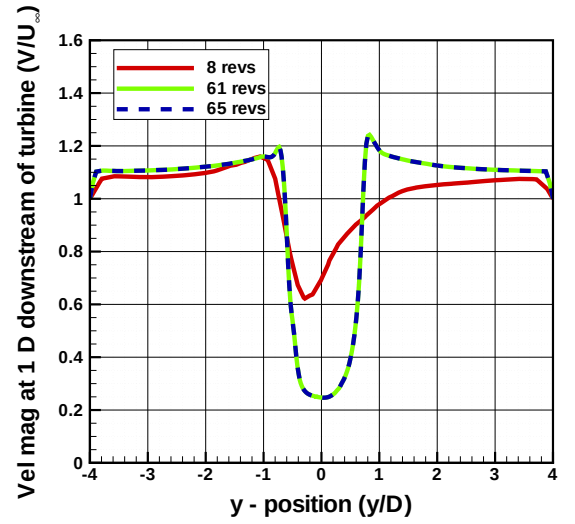
Figure 5.4: Convergence of blade torque and turbine power histories for $b = 12.5\%$ and $\lambda = 2$

For the simulations, where $b = 12.5\%$, about three times as many cycles needed to be completed to attain a statistically converged solution than for the higher blockage computations at $b = 50\%$. As illustrated in Figure 5.4 & 5.5, for $b = 12.5\%$ and $\lambda = 2$, convergence was reached after 61 revolutions. As may be anticipated from Figure 5.4, C_P remained unaltered to the second significant figure for 10 consecutive cycles after revolution 24, but the flow-field remained unconverged. As shown in Figure 5.5b - 5.5d, the velocity magnitude at the various cross-stream traverses examined were not considered converged until after revolution 61.

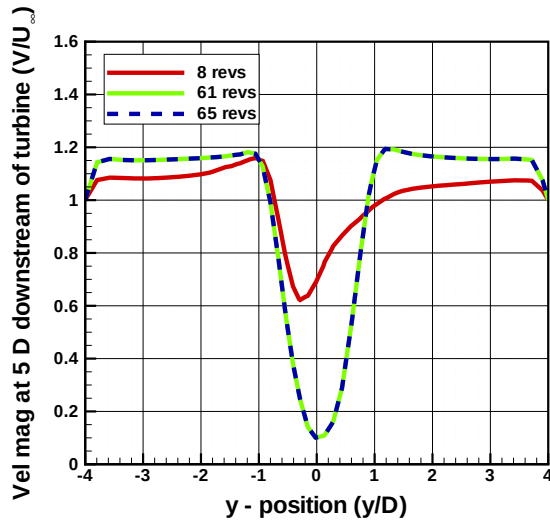
Moreover, it is observed that the initial vertical asymmetries apparent in Figure 5.5c & 5.5d largely disappear as the solution reaches convergence. These asymmetries are associated with a starting vortex which is washed downstream over time, rendering a more symmetric steady-state wake profile.



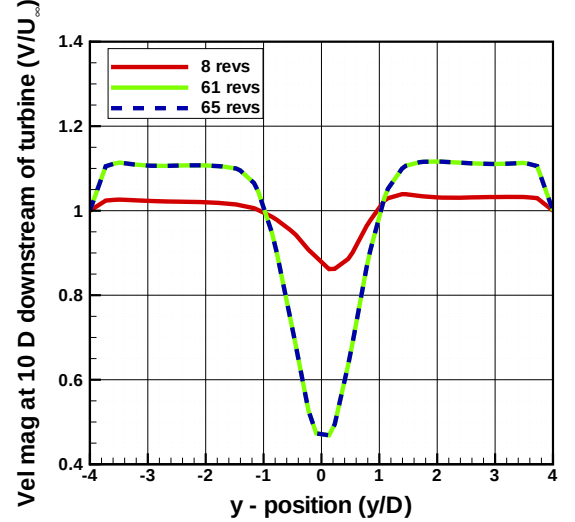
(a) Centreline velocity magnitude



(b) Velocity magnitude for a cross-stream traverse at 1 D downstream of turbine



(c) Velocity magnitude for a cross-stream traverse at 5 D downstream of turbine



(d) Velocity magnitude for a cross-stream traverse at 10 D downstream of turbine

Figure 5.5: Wake velocity profiles for $b = 12.5\%$ and $\lambda = 2$

The simulations conducted with $b = 6.25\%$ required the most revolutions of the four blockages tested to reach solution convergence. For $b = 6.25\%$ and $\lambda = 2$, C_P did not change to the second significant figure for 10 consecutive revolutions after having completed cycle 38. It took 33 further revolutions until the wake profiles examined remained unchanged for 5 cycles, see Figure 5.7.

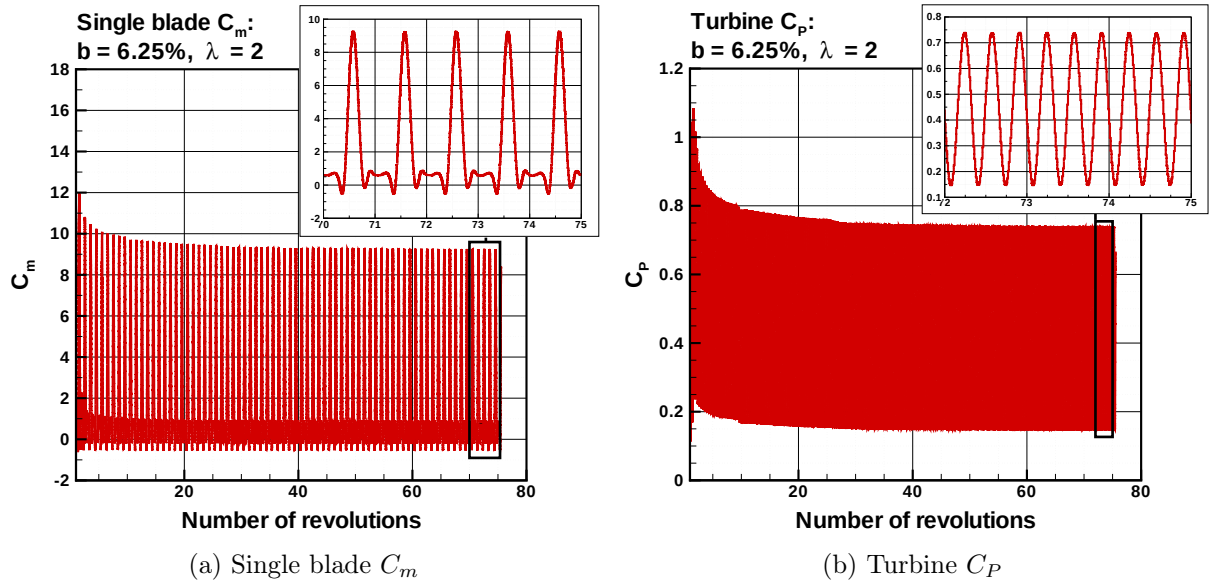
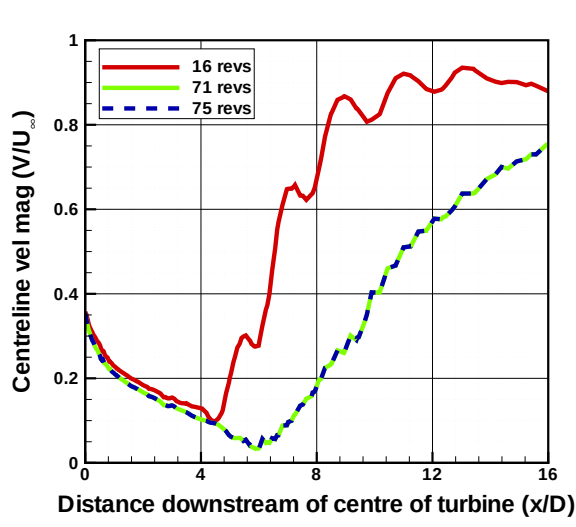


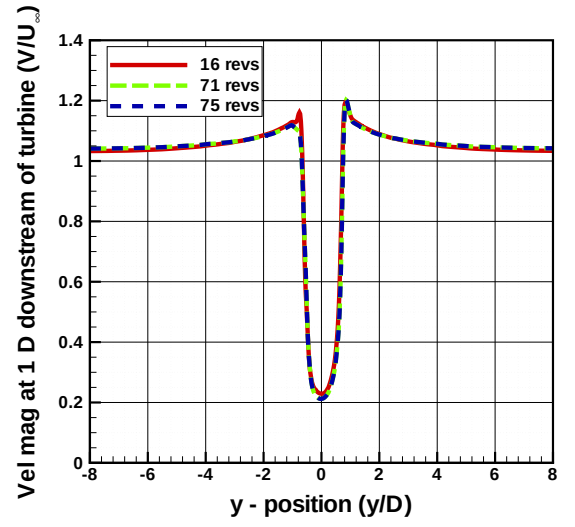
Figure 5.6: Convergence of blade torque and turbine power histories for $b = 6.25\%$ and $\lambda = 2$

Thus, at maximum power take-off, the simulations for the least blocked case took almost 3.5 as many cycles to reach convergence than for 50% blocked configuration. It seems that the more highly blocked cases are forced into a converged solution faster due to flow confinement effects.

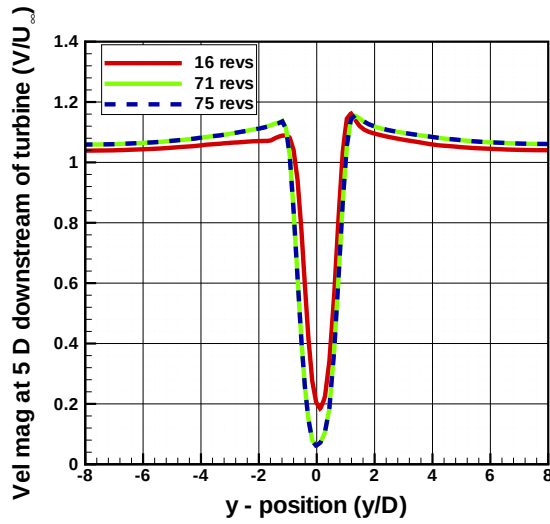
The asymmetry observed in some of the wake velocity plots after revolution 8 for $b = 12.5\%$ at $\lambda = 2$, is also apparent for $b = 6.25\%$ at $\lambda = 2$ after 16 revolutions, see Figure 5.7c & 5.7d. As before, the asymmetry largely disappears as the solution reaches convergence.



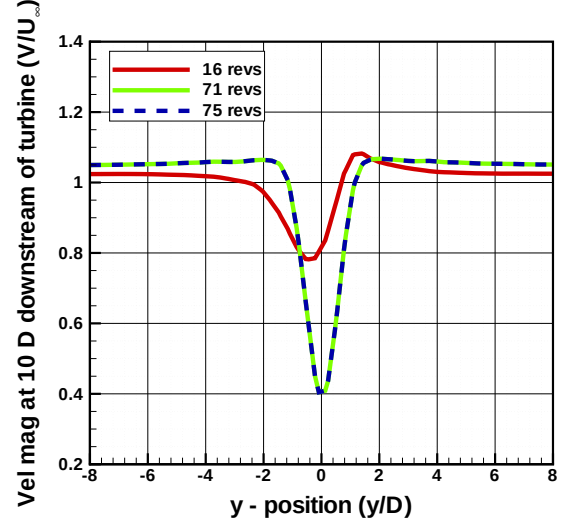
(a) Centreline velocity magnitude



(b) Velocity magnitude for a cross-stream traverse at 1 D downstream of turbine



(c) Velocity magnitude for a cross-stream traverse at 5 D downstream of turbine



(d) Velocity magnitude for a cross-stream traverse at 10 D downstream of turbine

Figure 5.7: Wake velocity profiles for $b = 6.25\%$ and $\lambda = 2$

Figure 5.8 shows the number of revolutions required until C_P remained unchanged for 10 consecutive revolutions, for all tip speed ratios simulated for the four different blockages considered.

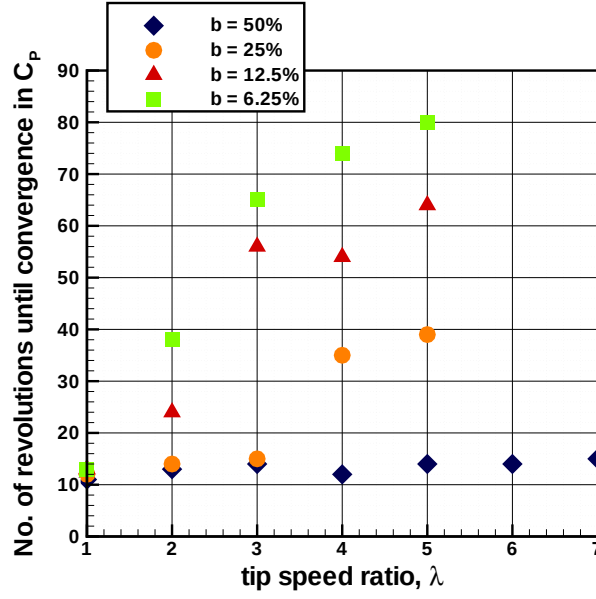


Figure 5.8: Number of revolutions required for convergence in C_P

Figure 5.8 indicates that the observations made above based on the tip speed ratio for maximum C_P , λ_{opt} , hold for the entire range of tip speed ratios simulated; i.e. that longer is required to achieve convergence as blockage is decreased. This is likely due to the increased time required for mixing between core and bypass flows as blockage is decreased and physical wake width is permitted to increase; see Figure 5.3d, 5.5d & 5.7d, which indicate the growth in wake width as blockage is reduced.

Moreover, it is apparent that the number of revolutions until convergence is obtained is also dependent on tip speed ratio, λ . In particular for $b = 6.25\%$, 12.5% & 25% , the higher λ , the more revolutions are required until solution convergence. This is to be expected, as for a fixed free-stream velocity, a decrease in the rotational velocity, ω , associated with a decrease in λ , means that a larger number of revolutions is required for free convection of a particle through the entire domain in the streamwise direction.

5.1.2 Results

Figure 5.9 shows the power and thrust curves simulated for the 3-bladed turbine with a turbine solidity of 0.125 and the four blockage ratios, b , considered of 6.25%, 12.5%, 25% and 50%.

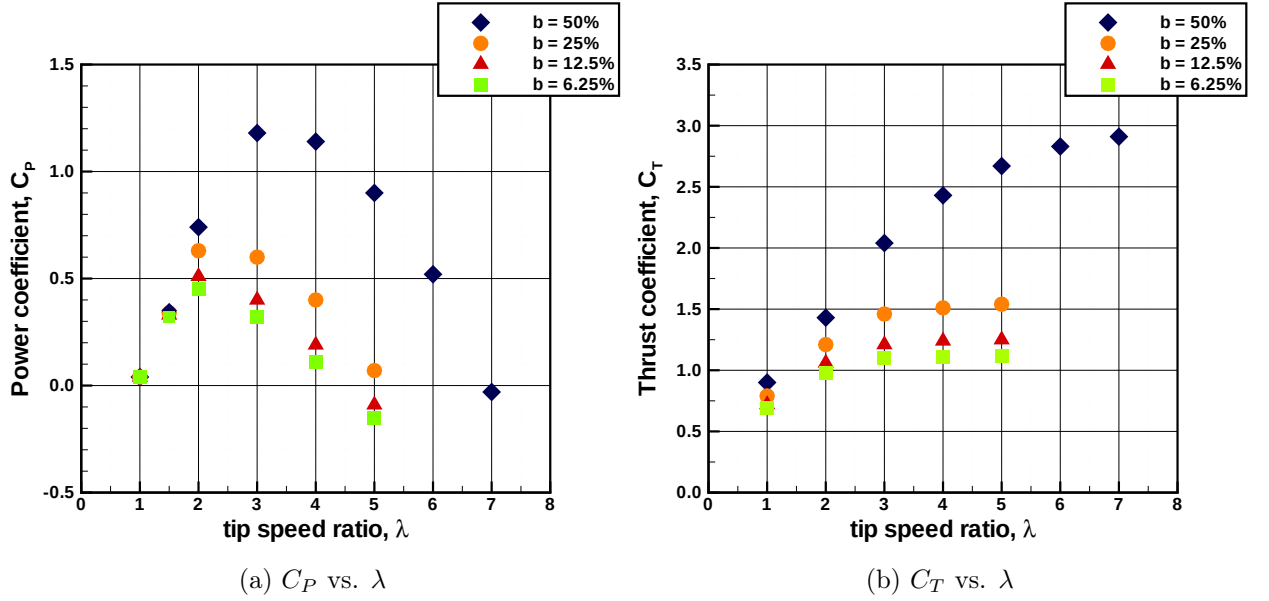


Figure 5.9: Power and thrust coefficient variation for varying blockage, b

When studying the various C_P curves in Figure 5.9a, three observations may be made:

1. As already discussed in Section 4.1.2, all power curves take the expected bell shape.
2. Testing the same turbine configuration, but changing the domain width and thus the turbine's effective blockage can have a very significant effect on the power produced by the turbine; max $C_P = 1.18$ for the 50% blocked case, against max $C_P = 0.63$ for the 25% blocked case, max $C_P = 0.54$ for the 12.5% blocked case and max $C_P = 0.45$ for the lowest blocked case with $b = 6.25\%$.
3. The higher the blockage, the greater the width of the power curve and the higher

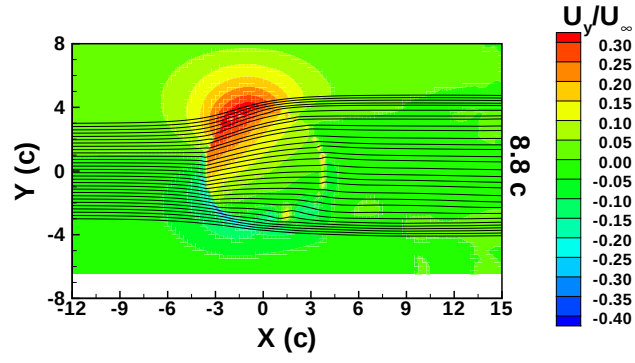
the maximum tip speed ratio at which positive power occurs. Also, note that $\max C_P$ occurs at higher λ with increasing blockage.

These differences can be explained by considering that the optimum design configuration lies between two extremes; a turbine that presents a very high and a very low flow resistance. On the one hand, the power output is proportional to the streamwise thrust that the turbine exerts on the flow. However, the larger the thrust, the larger the flow resistance and hence the lower the mass flow and kinetic energy flux through the turbine. In order that the turbine presents an “efficient” flow resistance to the flow, the right balance of solidity, tip speed ratio and flow confinement is sought.

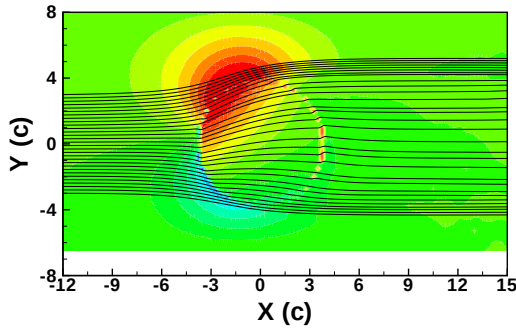
5.1.2.1 Time-averaged flow fields

The differences in streamwise thrust exerted by each turbine configuration are shown in Figure 5.9b.

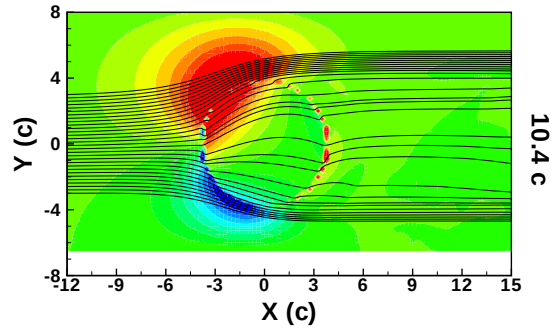
Firstly, it is noted for all four blockages that, as expected, C_T increases, as λ is increased. This may also be concluded from Figure 5.10, which shows the time-averaged flow fields at three different tip speed ratios, $\lambda = 2, 3$ & 6 , for $b = 50\%$. On these plots streamlines have been added in order to facilitate visualisation of the streamwise velocity induction factor, β . The cross-stream width of the streamtubes was fixed to be 6 chord lengths, c , at $12 c$ upstream of the centre of the turbine. By comparison of the expansion of the streamtubes, it is evident that the highest tip speed ratio, see Figure 5.10c, results in the widest downstream streamtube implying the greatest flow impedance. However, the power take-off at $\lambda = 6$ is only 44% of the power generated at $\lambda = 3$, which is partly due to the lower mass flux through the turbine at the higher tip speed ratio associated with the significantly higher impedance presented by the turbine.



(a) $b = 50\%$, $\lambda = 2$, $C_P = 0.74$



(b) $b = 50\%$, $\lambda = 3$, $C_P = 1.18$



(c) $b = 50\%$, $\lambda = 6$, $C_P = 0.52$

Figure 5.10: Time-averaged flow fields for different λ at blockage, $b = 50\%$, together with instantaneous streamlines

Secondly, Figure 5.9b shows for all λ that the higher the blockage, the larger C_T ; for example, at $\lambda = 3$:

- for $b = 50\%$, $C_T = 2.04$,
- for $b = 25\%$, $C_T = 1.46$,
- for $b = 12.5\%$, $C_T = 1.21$ and
- for $b = 6.25\%$, $C_T = 1.10$.

This illustrates that the larger power take-off, which is proportional to the streamwise thrust as well as the streamwise flow velocity, observed for the higher blockages is due to both an increase in C_T as well as an increase in the local streamwise flow velocity, U_x , which is discussed below.

Figure 5.11 shows the time-averaged flow-fields at λ corresponding to max power, λ_{opt} , for three different blockages, $b = 6.25\%$, $b = 12.5\%$ & $b = 50\%$. By comparison of the expansion of the streamtubes, it is evident that the lowest blockage, see Figure 5.11c, results in the widest downstream streamtube implying the greatest relative flow impedance; the streamtube increases by a factor of 2.3 for $b = 6.25\%$ at $\lambda = 2$, by a factor of 2.1 for $b = 12.5\%$ at $\lambda = 2$ and 1.6 for $b = 50\%$ at $\lambda = 3$.

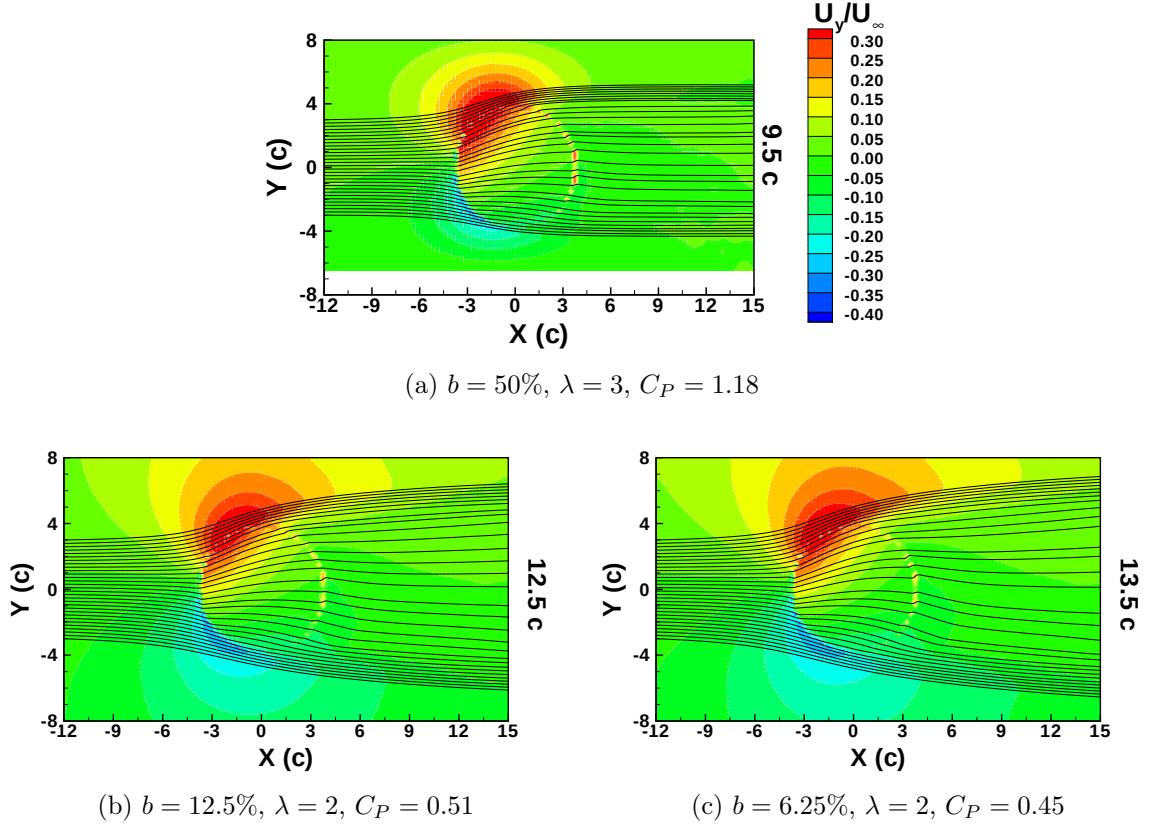


Figure 5.11: Time-averaged flow fields at λ_{opt} for different blockages together with instantaneous streamlines

The free-stream velocity upstream of the turbine is lowest for the least blocked case, see Figure 5.19a, as the flow acceleration due to flow confinement effects is smaller than for a highly blocked case. Furthermore, as noted above, the lower the blockage, the higher the relative streamwise flow impedance, which implies that the streamwise flow velocity through and downstream of the turbine is lower; this is further illustrated

below in Figure 5.19a, which shows a comparison of the centreline velocity magnitude plots for the different blockages at λ_{opt} .

A change in the streamwise velocity has two effects:

1. it influences the mass flux through the turbine and
2. it influences the range of incidences the blades are presented with through the revolution cycle.

The change in the range of effective angle of attack due to the change in the streamwise velocity underlies the shift of the power curve to a larger range of λ for higher blockage ratios. The lower flow impedance and higher streamwise flow velocities observed for the highest blockage case, see Figure 5.11a, act to increase the angle of incidence, as apparent from Figure 4.16. At low λ , this results in a decrease in the power take-off due to blade stall, whilst at high λ , it results in an increase in C_P due to increased blade incidence, and hence an overall shift of the power curve to higher λ .

The increase in mass flux associated with the increase in streamwise flow velocity is important, when examining the effect of blockage on power produced by the turbine. The significant increase in max C_P for the 50% blocked case observed above may be attributed to two factors arising from an increase in blockage:

1. increase in mass flux,
2. increase in the available pressure drop.

As discussed above, the reduction of the distance from the centre of the turbine to the cross-stream boundaries has resulted in significant flow confinement, i.e. in an increase in the streamwise flow velocity and thus an increase in the mass flux flowing through the turbine. The results from the numerical simulations suggest that at peak power the average mass flow rate passing through the turbine is 1.4 times larger for the 50%

blocked case than for the 6.25% blocked case, which indicates a significant increase in the kinetic energy flux flowing through the turbine.

In addition, the potential for energy extraction is increased for an increase in blockage due to an increase in the available pressure drop across the turbine. This may be explained by considering the linear momentum actuator disc theory for a parallel-sided tube, discussed in Houlsby et al. (2008).

Due to an increase in blockage and thus an increase in flow confinement effects not only the flow velocity through the turbine, i.e. the core flow, increases, but also the flow velocity of the bypass. The bypass flow region may be treated as inviscid, which means that an increase in the bypass flow velocity results in a pressure drop. Downstream, where the bypass and core flows are parallel, the pressure must be constant across the streamtubes, as they are parallel, and hence flow confinement pushes the downstream pressure downwards and thus enabling a larger change in pressure across the turbine.

This means that the potential for energy extraction is significantly larger, the higher the blockage. The increase in kinetic energy flux flowing through the turbine as well as the increase in available pressure drop across a turbine due to flow confinement, allow the Betz limit, see Burton et al. (2001), for unconfined flow conditions, $C_{P_{\max}} = 0.59$, to be greatly exceeded; $C_P = 1.18$ for $b = 50\%$ and $\lambda = 3$.

5.1.2.2 Sectional lift and drag forces

In order to examine how the (additional) energy is extracted, the sectional lift and drag forces experienced by a single blade of the different turbines at $\lambda = 3$ & 4, as it traverses a full rotation cycle, were studied, see Figure 5.12 & 5.13. The lift and drag coefficients, defined in Equation 5.1.3 & 5.1.4, were computed following the procedure outlined in Section 4.1.2.4;

$$C_L = \frac{L}{\frac{1}{2}\rho U_r^2 c} \quad (5.1.3)$$

$$C_D = \frac{D}{\frac{1}{2}\rho U_r^2 c} \quad (5.1.4)$$

where L and D are the lift and drag per unit span and the resultant flow velocity, U_r , is given by Equation 5.1.5 :

$$U_r = \sqrt{(U_x + U_\theta \cos \theta)^2 + (U_y + U_\theta \sin \theta)^2} \quad (5.1.5)$$

where U_y and U_θ are the cross-stream and blade rotational velocities respectively and θ the azimuth angle.

In the manner plotted, positive incidence refers to the downstream pass of the blade, whilst negative incidence refers to the upstream blade pass. The markers 1, 2, 3 and 4 indicate the azimuthal positions of the start of the 1st, 2nd, 3rd and 4th quadrants at $\theta = 0^\circ$, 90° , 180° and 270° respectively.

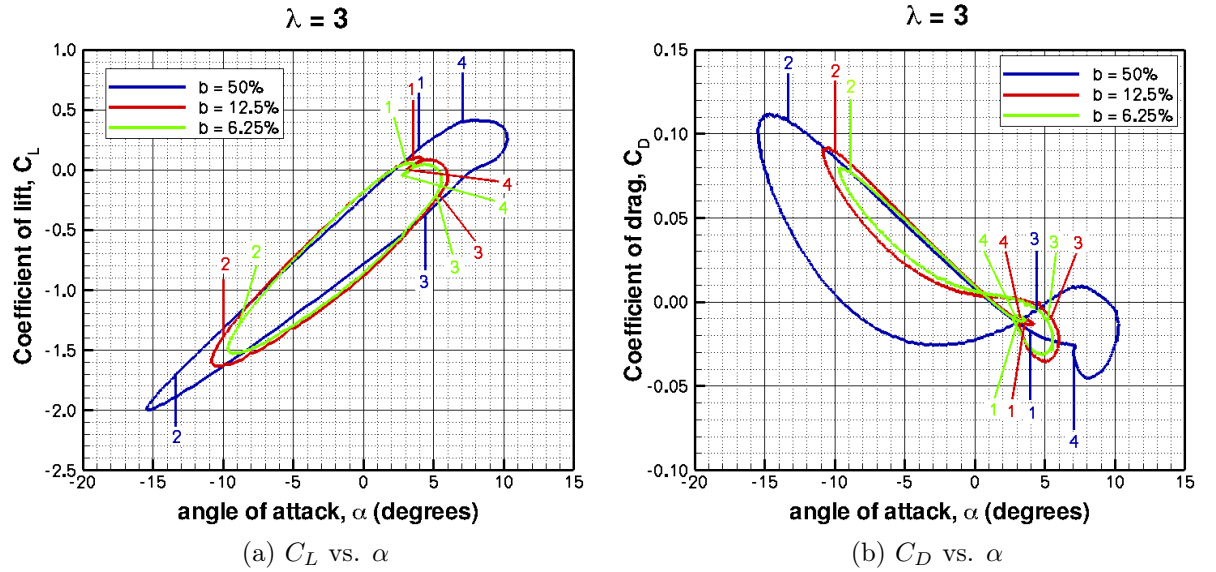


Figure 5.12: Blade coefficient histories at $\lambda = 3$

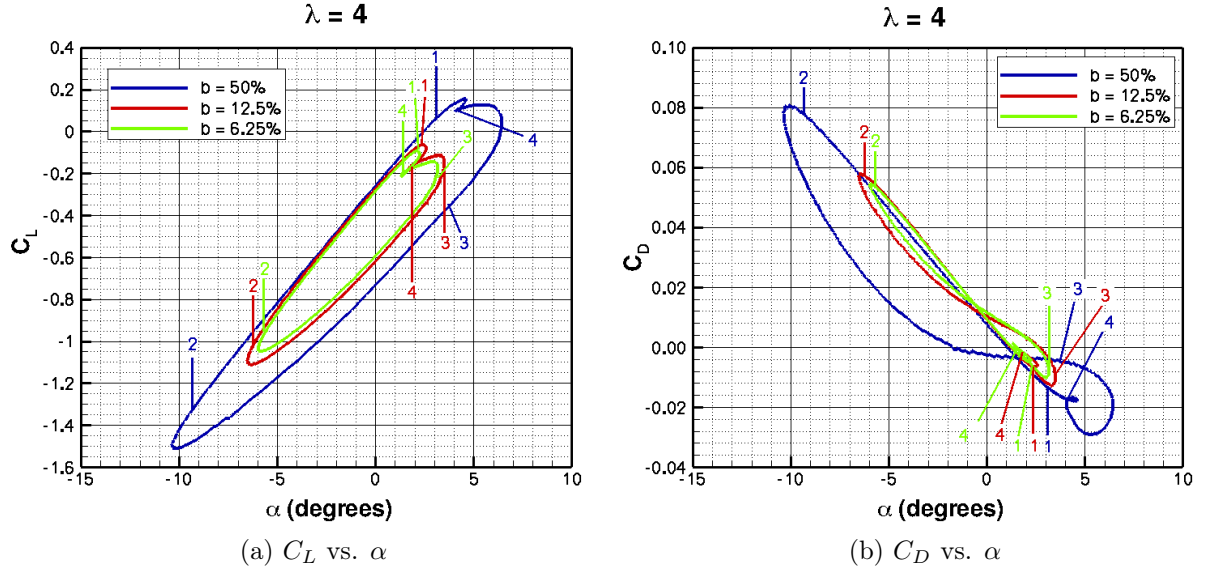


Figure 5.13: Blade coefficient histories at $\lambda = 4$

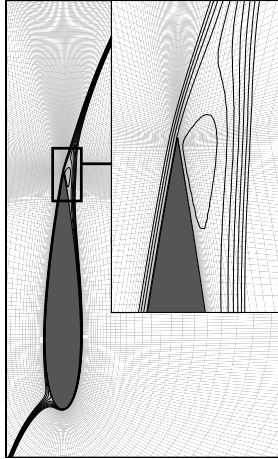
Figure 5.12a & 5.13a show that the shapes of the corresponding lift curves for $b = 6.25\%$, 12.5% & 50% are similar. As discussed in Chapter 4, the typical C_L plot for a blade of a cross-flow turbine exhibits a hysteresis loop, which is a result of shed vorticity and its effect on the incident flow-field. The loop leads to a larger C_L for the same angle of attack when incidence is decreasing than when it is increasing. Hence a larger C_L is expected in quadrants 2 & 4 relative to that expected in quadrants 1 & 3.

The principal difference between the C_L traces arises from the different streamwise flow velocities observed for the various blockages. From these plots it is apparent, as discussed above, that an increase in the local streamwise flow velocity, due to an increase in effective blockage, acts to increase blade incidence, α ; at $\lambda = 3$:

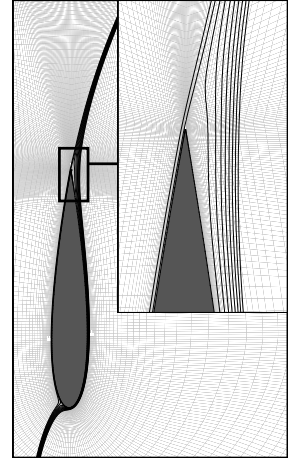
- $\alpha_{\max} = (-) 15.5^\circ$ for $b = 50\%$,
- $\alpha_{\max} = (-) 10.9^\circ$ for $b = 12.5\%$ and
- $\alpha_{\max} = (-) 9.7^\circ$ for $b = 6.25\%$.

As $\alpha_{\max}$ is larger for $b = 50\%$ than for the lower blockages, $\max C_L$ is also largest for the highest blockage, resulting in higher torque and thus power. Moreover, the larger the maximum angle of attack reached, which implies more intense shedding of bound vorticity, the wider the lift force loops generally are. This implies that as blockage increases C_L will be larger for the same α in quadrants 2, 3 & 4, due to achieving a higher $\alpha_{\max}$ on the upstream passage and hence vorticity shed.

This is not the case, however, for $b = 50\%$ and $\lambda = 3$, which exhibits the thinnest loop despite reaching the largest angles of attack. This is because $\alpha_{\max}$ for $b = 50\%$ and $\lambda = 3$ is closer to the stall angle and flow separation is initiated at the trailing edge, see Figure 5.14a, which shows an instantaneous streamline plot for $b = 50\%$ and $\lambda = 3$ at $\theta = 104.5^\circ$. In comparison to Figure 5.14b, which illustrates an instantaneous streamline plot for $b = 6.25\%$ and $\lambda = 3$ at $\theta = 106.0^\circ$, it is apparent that the separation point, initiated at the trailing edge, has moved up the blade of the more highly blocked case. However, from the lift curve, it may be deduced that the separation is not so severe as to cause complete blade stall but there is loss in lift through much of quadrant 2 relative to the lower blockage cases for which stall was avoided.



(a) $b = 50\%$, $\lambda = 3$, $\theta = 104.5^\circ$



(b) $b = 6.25\%$, $\lambda = 3$, $\theta = 106.0^\circ$

Figure 5.14: Streamline plots at $\theta \approx 105.0^\circ$ for $\lambda = 3$ and $b = 50\%$ & 6.25% . Note that to ease comparison between cases, each flow field has been rotated such that the blade is traversing vertically downward.

In conclusion, as long as blade stall is not reached, the increase in $\alpha_{\max}$ observed for an increase in blockage results in an increase in $\max C_L$ as well as an increase in C_L for a given angle of attack for the majority of quadrants 2, 3 & 4. This partly explains the increase in power take-off observed for an increase in blockage.

In Chapter 3, it was established that the torque per unit length of blade, Q , is given by Equation 3.1.6, and using the coefficients of lift and drag, as defined in Section 4.1.2.4, renders Equation 5.1.6:

$$Q = BR(L \sin \alpha - D \cos \alpha) = BR \frac{1}{2} \rho U_r^2 c (C_L \sin \alpha - C_D \cos \alpha) \quad (5.1.6)$$

From Equation 5.1.6, it is apparent that the second key contributor to the (positive) torque and thus the power generated is the resultant velocity, U_r , which is given by Equation 5.1.5.

Although the cross-stream velocity, U_y , decreases as blockage is increased, U_r still increases due to the larger increase in streamwise velocity, U_x , with increasing blockage. At $\lambda = 3$, the average of the resultant velocity over one cycle:

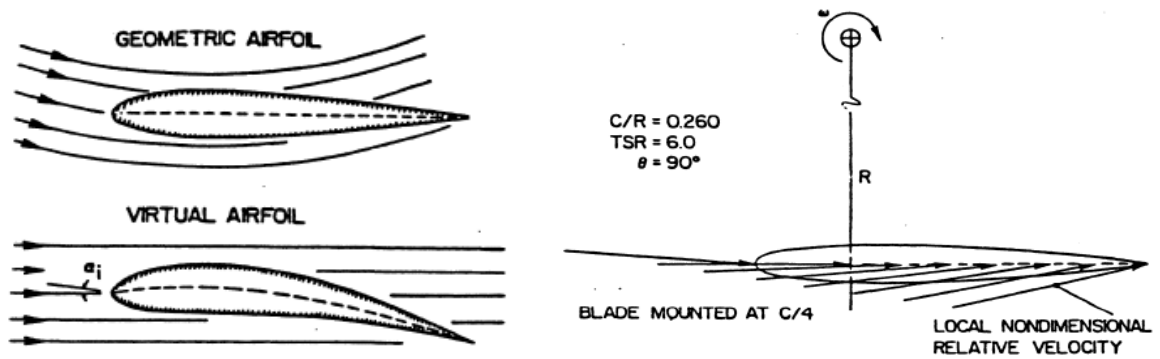
- $U_{r_ave}/U_\infty = 1.30$ for $b = 50\%$,
- $U_{r_ave}/U_\infty = 1.17$ for $b = 12.5\%$ and
- $U_{r_ave}/U_\infty = 1.14$ for $b = 6.25\%$.

This implies that the lift generated over once cycle and thus torque and power increase with blockage, because an increase in blockage results in an increase in both C_L as well as U_r (integrated over the revolution).

Furthermore, it is of interest to note that when comparing C_L at $\alpha = 0^\circ$ for $\lambda = 3$ & 4, see Figure 5.12a & 5.13a, C_L is similar for all three blockages and is non-zero; at $\alpha = 0^\circ$:

- $C_L \approx (-) 0.2$ at $\lambda = 3$,
- $C_L \approx (-) 0.25$ at $\lambda = 4$.

This is contrary to expectations, as one might expect $C_L = 0$ at $\alpha = 0^\circ$ for a straight blade; this pseudo-camber is because of the curved path of the blades arising from the turbine's circular rotation. It has been shown that the hydrodynamic characteristics of a blade in a curvilinear flow-field differ from those of the same foil section in a rectilinear flow, see Migliore et al. (1980). The effects of a curvilinear flow-field can be examined with the help of conformal mapping. As evident from Figure 5.15, the transformation affects a change in effective camber, see Figure 5.15a, and introduces an effective angle of incidence, see Figure 5.15b, in addition to the blade's geometric angle of attack.



(a) Virtual camber & incidence, taken from Migliore et al. (1980) (b) Chordwise variation in α , taken from Migliore et al. (1980)

Figure 5.15: Illustration of effect of curved path of cross-flow turbine blades

In other words, a symmetric blade with zero pitch angle travelling in a curved path behaves like a similar but cambered foil with a non-zero pitch angle that moves in a straight line; this explains the origin of the off-set of the lift curves observed in Figure 5.12a & 5.13a. Such an offset was not observed for the lift curves computed for the blades of the turbines discussed in Chapter 4, because the turbine solidities were significantly smaller, so that the effect of the (virtual) camber became negligible; i.e.

as c/R was small the flow field encountered by the rotating blade could be accurately approximated to be rectilinear.

Also, when comparing the maximum blade incidences on the upstream and downstream passes, it is apparent that $\alpha_{\max}$ is significantly lower on the latter, see Figure 5.12 & 5.13. This is due to the streamwise flow resistance presented by the turbine, which results in lower local U_x on the downstream side and thus a reduction in α . Regarding the differences in $\alpha_{\max}$ between upstream and downstream passes for the different blockages, for $\lambda = 3$, $\alpha_{\max}$ is reduced by 33.5% for $b = 50\%$, by 44.4% for $b = 12.5\%$ and 43.0% for $b = 6.25\%$. This concurs with the interpretation of the time-averaged flow fields in Figure 5.11 that the relative effect of flow impedance is lowest for the highest blocked case.

Moreover, a comparison of $\alpha_{\max}$ on both upstream and downstream passes for $\lambda = 3$ & 4 shows that the computations for the higher λ lead to lower $\alpha_{\max}$. This is, as discussed above, because of the higher flow resistance associated with an increase in λ , which results in a reduction in α .

The drag plots in Figure 5.12b & 5.13b exhibit three similar curves for $b = 6.25\%$, 12.5% & 50% in terms of their shape. As for the lift, the drag force hysteresis loop increases in width with an increase in $\alpha_{\max}$ associated with an increase in blockage.

Moreover, it is noted that the drag plots show negative C_D values. (Averaging) errors might have been introduced by computing the forces, velocities and angles of attack from time-averaged flow-fields. As C_D is small in magnitude, small errors in computed incidence may be of relative significance resulting in (unphysical) negative drag forces.

Further, negative C_D occurs when the blade is in retreat or on the downstream passage. In all cases the blade passes through a wake region, for which the time-averaged flow statistics may not provide an appropriate representation and the computed instantaneous incidences will suffer greater error than these computed on the upstream passage.

Furthermore, it is known that, for a rotating high solidity blade, effective incidence varies with chord, see Figure 5.15b, and hence the computed incidence may be an overly simplistic flow quantifier, rendering the computed incidences only approximate. It is suggested to treat the drag results with caution.

5.1.2.3 Blade torque

Figure 5.16 shows the torque coefficient, C_m , for a single blade, as it completes a revolution. As outlined in Section 3.1.2, $\theta = 0^\circ$ corresponds to a blade's top position vertically above the centre of the turbine. Figure 5.16a shows a comparison for the blockages, $b = 6.25\%$, 12.5% & 50% , at the tip speed ratio of maximum power take-off, λ_{opt} . Figure 5.16b shows a similar comparisons but λ is fixed at 2.

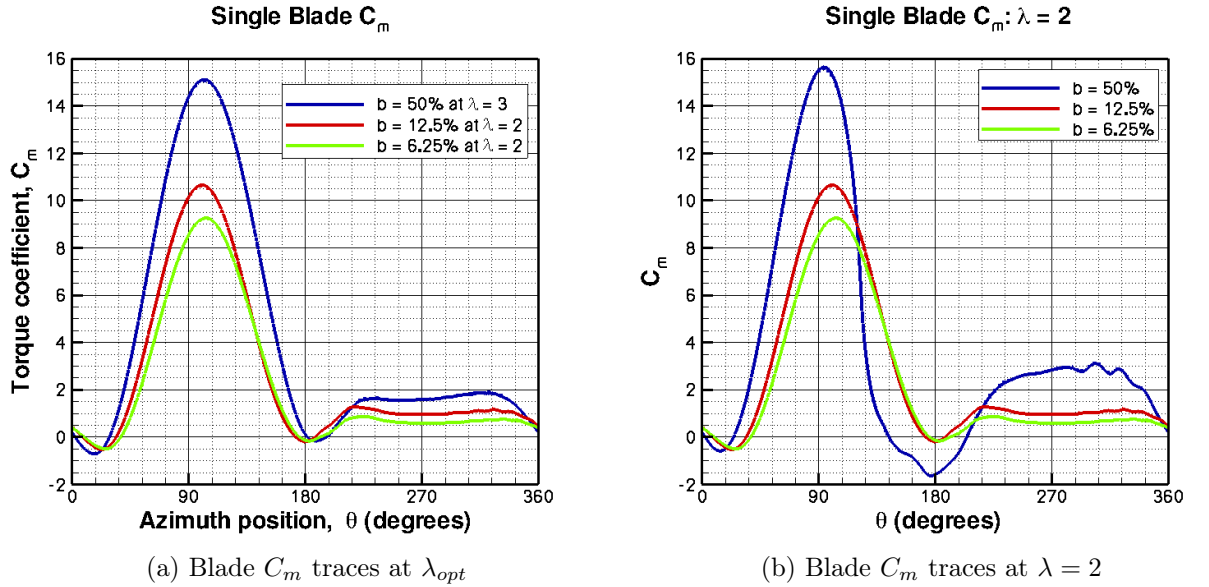


Figure 5.16: Comparisons of blade torque coefficient, C_m , histories

All plots in Figure 5.16a & 5.16b exhibit the expected shape for a single blade torque trace. As the blade reaches $\theta \approx 90^\circ$ and thus maximum angle of incidence, the first peak in the torque trace is observed; at around $\theta = 180^\circ$ the torque trace reaches a minimum, when the blade is in retreat at the bottom of the turbine. As to the downstream side, it

is evident that the blades contribute little to no positive torque, as they operate under perturbed flow conditions. In fact, the lower the effective blockage, the less (positive) torque is provided by the blades on the downstream passage, as the local streamwise velocities are reduced due to lower flow confinement effects encountered with lower blockage.

Differences in C_m arising from the different blockages may also be observed for the upstream blade passes. When studying the C_m plots in Figure 5.16, it is apparent that the larger the blockage, the larger $\max C_m$, which results in the larger C_P values computed for the more highly blocked cases, as discussed above.

From Figure 5.16b it may be deduced that the blade of the most highly blocked turbine has undergone a severe stalling event at $\lambda = 2$ in the second quarter of the cycle, indicated by the asymmetry of the upstream passage torque; also, C_m per blade is atypically lower for $b = 50\%$ than for the lower blockages for $120^\circ < \theta < 220^\circ$. To examine this further, instantaneous streamline plots have been studied for the second quadrant for $b = 50\%$ and $b = 12.5\%$ at $\lambda = 2$, see Figure 5.17.

5.1.2.4 Instantaneous streamline plots

The instantaneous streamline plots for $b = 50\%$ and $\lambda = 2$ at $120^\circ < \theta < 220^\circ$, see Figure 5.17a - 5.17c, confirm the conclusion drawn from the torque traces that the blades have experienced blade stall in the second quadrant. These figures show that the flow is separated over the entire length of the (inner) suction surface over the range of azimuth shown. Moreover, large scale vortex shedding associated with flow separation may be observed.

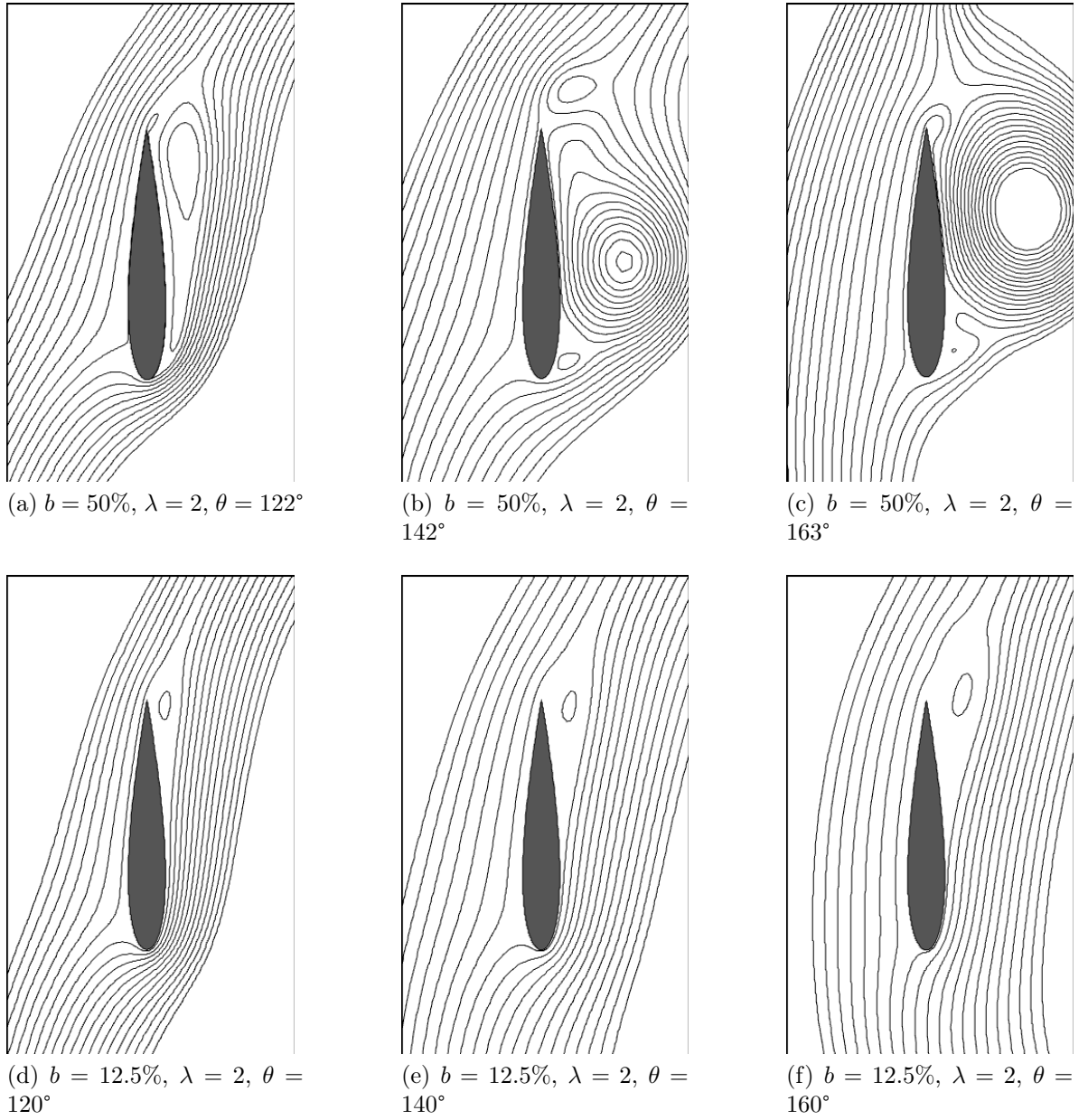


Figure 5.17: Instantaneous streamline plots for blockages, $b = 50\%$ & $b = 12.5\%$ at $\lambda = 2$ for $120^\circ < \theta < 163^\circ$. Note that to ease comparison between cases, each flow field has been rotated such that the blade is traversing vertically downward.

For $b = 12.5\%$, $\lambda = 2$ at $120^\circ < \theta < 220^\circ$, see Figure 5.17d - 5.17f, it is apparent that flow separation is initiated at the trailing edge, but the flow remains attached at the leading edge for $120^\circ < \theta < 220^\circ$ and for a good portion of the blade chord. This results in less significant vortex structures in the lower blockage case at this tip speed ratio.

5.1.2.5 Turbine torque

Figure 5.18 shows comparisons of the turbine torque (net of all blade torques) C_m histories for λ_{opt} as well as for fixed $\lambda = 2$. The observations are very similar to those made above for the corresponding traces of a single blade. The higher the blockage, the higher the mean C_m ; also, the turbine C_m history for $b = 50\%$ & $\lambda = 2$ is asymmetric, indicating blade stall, as discussed above.

Moreover, turbine fatigue characteristics are expected to be essential to identify optimal designs for cross-flow turbines and, hence, it is important to note the more uniform loading experienced by the turbines with $b = 6.25\%$ and 12.5% than with the highest blockage; at λ_{opt} the difference in maximum and minimum C_m , ΔC_m , is 20% lower for $b = 6.25\%$ than for $b = 50\%$. Other factors influencing the smoothness of turbine torque are turbine solidity and the number of blades, as discussed in Section 4.1.2.5.

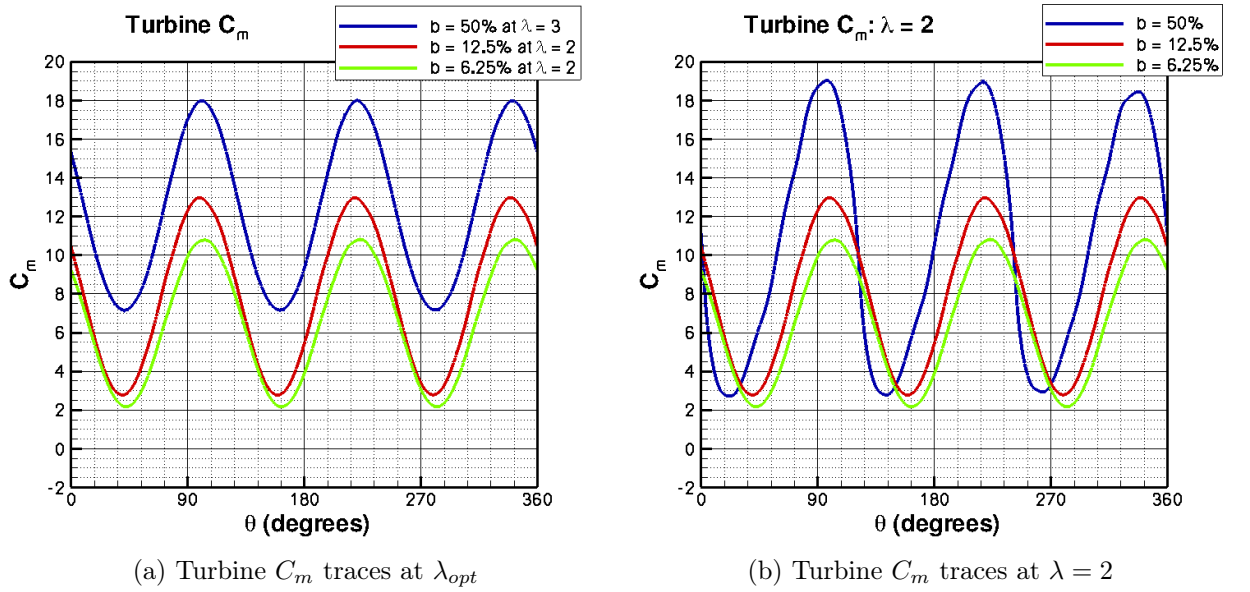
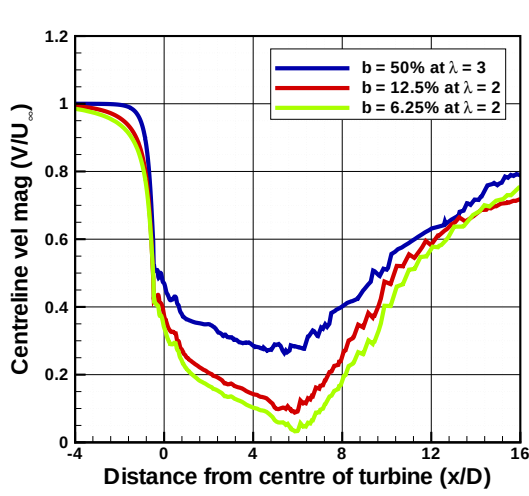


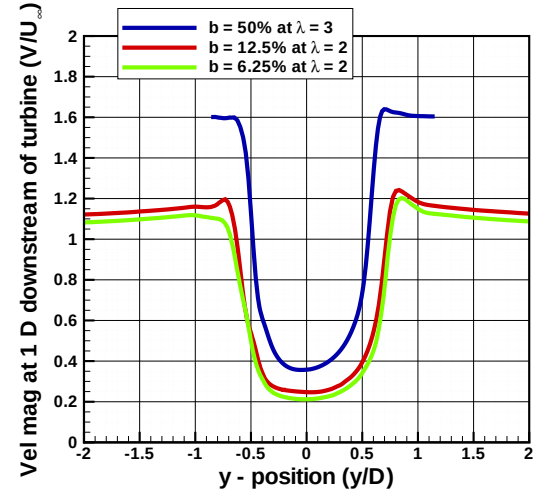
Figure 5.18: Comparisons of turbine C_m traces

5.1.2.6 Turbine wake

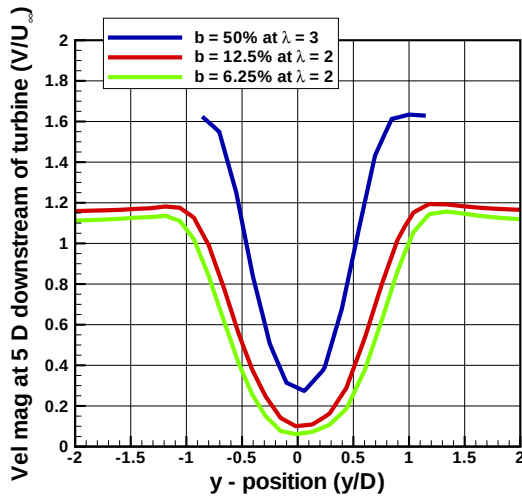
Furthermore, (wake) velocity plots have been compared for the blockages, $b = 6.25\%$, $b = 12.5\%$ and $b = 50\%$, at λ_{opt} , see Figure 5.19. As for the solution convergence investigation, see Section 5.1.1, four velocity plots have been examined; the centreline velocity magnitude as well as the velocity magnitude at cross-stream traverses at $1 D$, $5 D$ and $10 D$ downstream of the turbine centre.



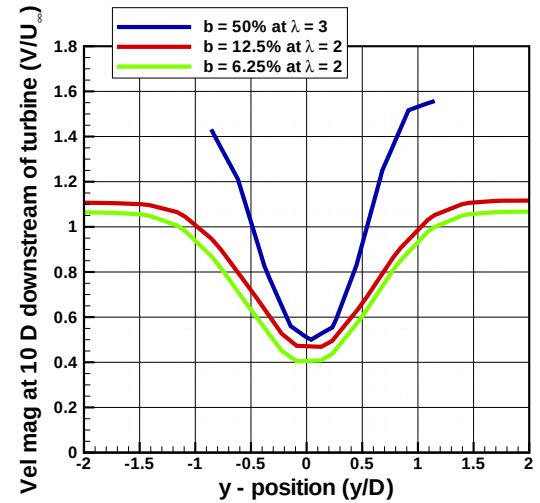
(a) Centreline velocity magnitude



(b) Velocity magnitude for a cross-stream traverse at $1 D$ downstream of turbine



(c) Velocity magnitude for a cross-stream traverse at $5 D$ downstream of turbine



(d) Velocity magnitude for a cross-stream traverse at $10 D$ downstream of turbine

Figure 5.19: Turbine wake profiles at maximum power point, λ_{opt}

Figure 5.19a shows the centreline velocity magnitude and it is apparent that the velocity magnitude, $V/U_\infty = 1$, at the inlet for all three blockages, as set by the corresponding boundary condition. Further downstream, as the turbine is approached, V/U_∞ reduces, which indicates the effect of the flow resistance presented by the turbine.

From the time-averaged flow fields plotted in Figure 5.11 it was concluded that the lower the blockage the greater the flow resistance presented by the turbine. This is confirmed by the plots in Figure 5.19a, which show that the reduction in the centreline velocity magnitude in and downstream of the turbine is more significant, the lower the blockage; at the centre of the turbine, the simulation for the turbine with $b = 50\%$ computed a velocity magnitude, V/U_∞ , of 0.47, whilst for the turbine with $b = 6.25\%$ a velocity magnitude, V/U_∞ , of 0.34.

Moreover, the reduction in the centreline velocity magnitude from the centre of the turbine to the point downstream of minimum velocity magnitude was computed to be 88% for $b = 6.25\%$, whilst only 40% for $b = 50\%$.

This is also why the plots of the velocity magnitudes at the cross-stream traverses at $1 D$, $5 D$ and $10 D$ downstream of the centre of the turbine, see Figure 5.19b - 5.19d, exhibit lower velocities, the lower the blockage. There is a corresponding inverse relationship to the velocity in the bypass region, which may be deduced by considering mass conservation. The plots for $b = 50\%$ show the narrowest wake widths in all three figures, which indicates that downstream of the turbine the flow remains (highly) accelerated for $b = 50\%$ due to flow confinement effects.

Figure 5.20 shows the centreline static, dynamic and total pressures for blockages, $b = 6.25\%$, 12.5% & 50% at λ_{opt} .

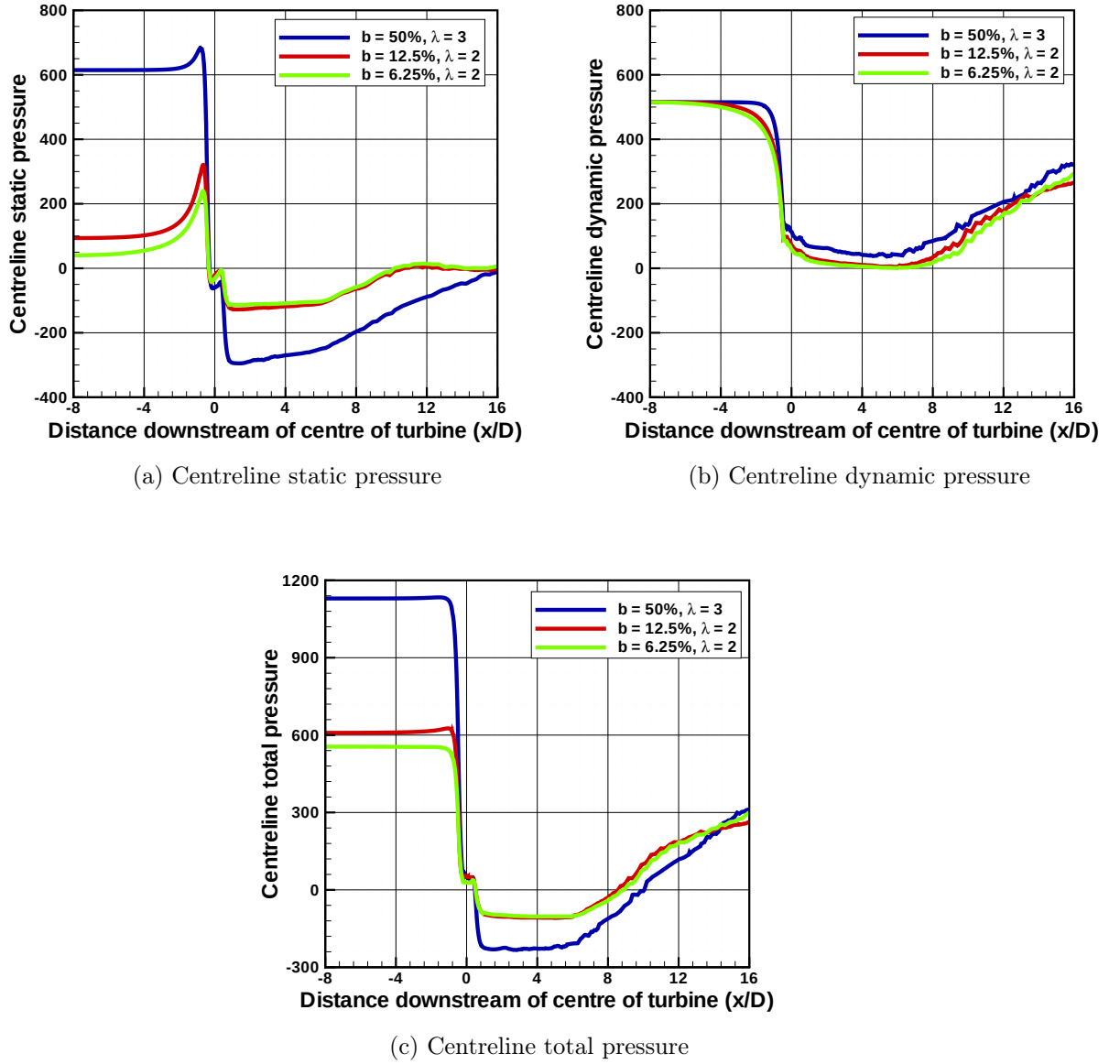


Figure 5.20: Comparisons of centreline wake pressures at maximum power point, λ_{opt}

From Figure 5.20a it may be deduced that the static pressure varies at the inlet depending on blockage. For the present (rigid lid) simulations, the energy is extracted from the domain in the form of pressure head, as is discussed in more detail in Section 5.2. The static pressure is set to 0 at the outlet, which implies that an increase in pressure head across the domain has to arise from an increase in the static pressure at the inlet; hence, the more power a turbine produces, which implies an increase in

the energy extracted from the domain, the larger is the static pressure required at the inlet to drive the flow through the domain. This explains why the static pressure is highest at the inlet for the most highly blocked case, which generates the largest C_P . The relationship between the power produced by a turbine and the energy extracted from the domain is discussed in more detail in Section 5.2.

Figure 5.20a shows that the turbine with the highest blockage leads to the highest drop in static pressure across the turbine; this concurs with the results in Figure 5.9b, which show that the turbine with $b = 50\%$ generates the highest thrust, which arises from the higher static pressure drop across the turbine relative to the pressure drop across the corresponding turbines with $b = 6.25\%$ & 12.5% .

Moreover, a comparison of the drop in static pressure, ΔP , across the front and rear passages shows that ΔP is greater on the upstream side for the three cases shown in Figure 5.20a; this implies a greater power generation on the upstream than on the downstream side, which ties in to the blade torque results discussed above.

Dynamic pressure, q , is defined, as:

$$q = \frac{1}{2}\rho|U|^2 \quad (5.1.7)$$

where ρ is the fluid's density and $|U|$ the velocity magnitude.

Hence, it is to be expected that the dynamic pressure plots are similar to the centreline velocity magnitude plots, compare Figure 5.19a & 5.20b. As for the velocity, the lower the blockage, the larger the drop in dynamic pressure across the turbine and the lower the minimum dynamic pressure downstream of the turbine.

As discussed in more detail in Section 5.2, the difference in total pressure, P_0 , between the inlet and outlet is the energy extracted from the domain, where P_0 is defined as:

$$P_0 = P + q = P + \frac{1}{2}\rho|U|^2 \quad (5.1.8)$$

where P and q are the static and dynamic pressures respectively.

The difference in the drop in total pressure across the turbine and domain for the different blockages illustrated in Figure 5.20c is an indicator of the difference in energy extracted from the flow by the different turbines, although we note that the total pressure may vary in the cross-stream direction. Hence, for $b = 50\%$, which generates more power than the turbines with $b = 6.25\%$ and $b = 12.5\%$, the drop in centreline total pressure is most significant.

5.1.2.7 Overall flow characteristics

In this section different visualisation options of the overall flow characteristics have been explored. Figure 5.21 and 5.22 show pressure and vorticity magnitude contour plots, respectively, at four different azimuthal positions, where vorticity, $\vec{\omega}$, is defined as the curl of the velocity field:

$$\vec{\omega} = \vec{\nabla} \times \vec{U} \quad (5.1.9)$$

As the flow-field at $\theta = 0^\circ$ resembles that at $\theta = 120^\circ$ and $\theta = 240^\circ$ for the 3-bladed turbine configuration simulated, the contour plots in Figure 5.21 and 5.22 are shown for an azimuthal range of $0^\circ < \theta < 120^\circ$.

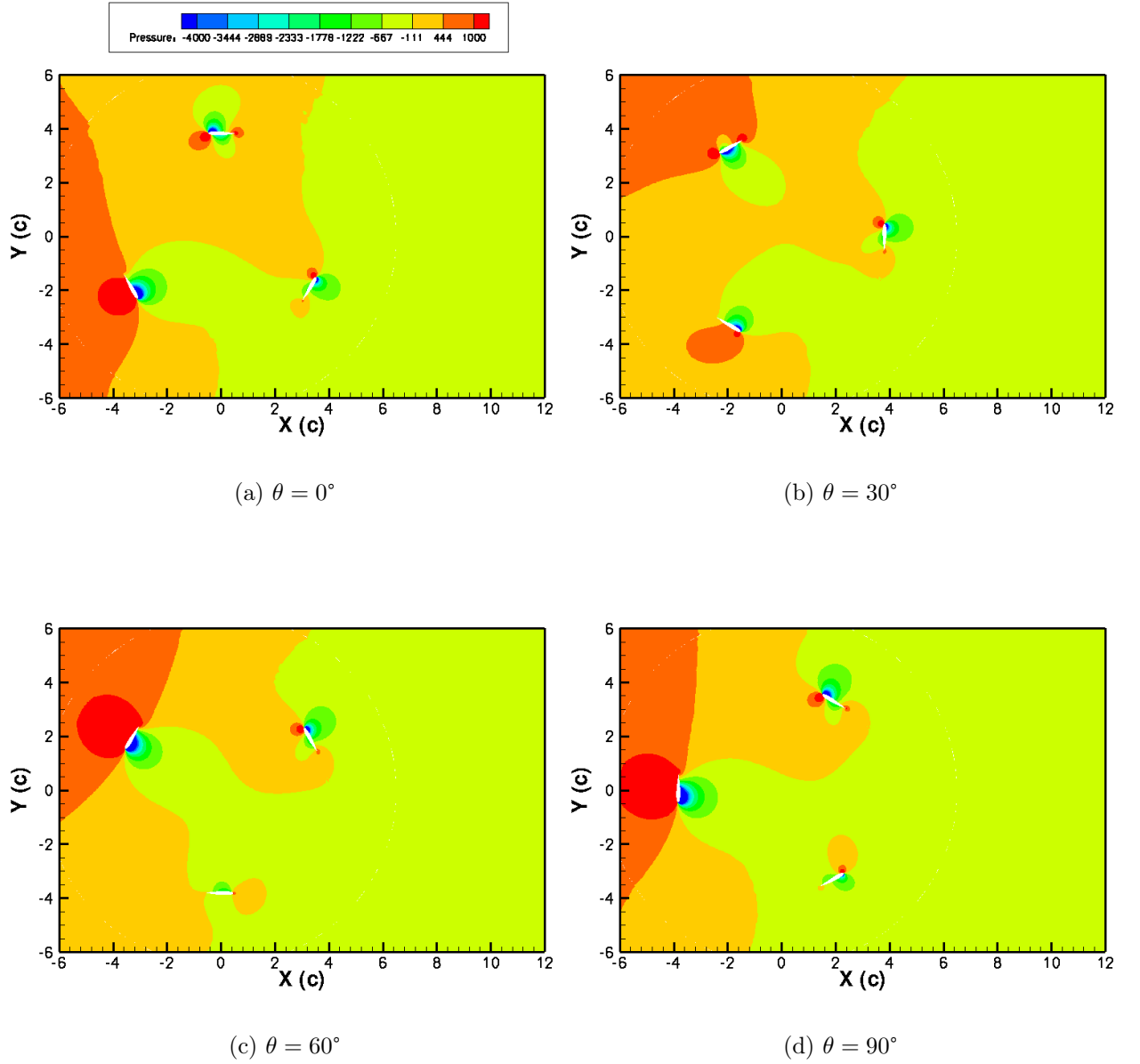


Figure 5.21: Pressure contour plots for various azimuthal positions, θ , where $b = 50\%$ and $\lambda = 3$

In the pressure contour plots in Figure 5.21 it can be noted that the suction and pressure surfaces switch between the upstream and downstream passages, as discussed in Section 1.3.2.

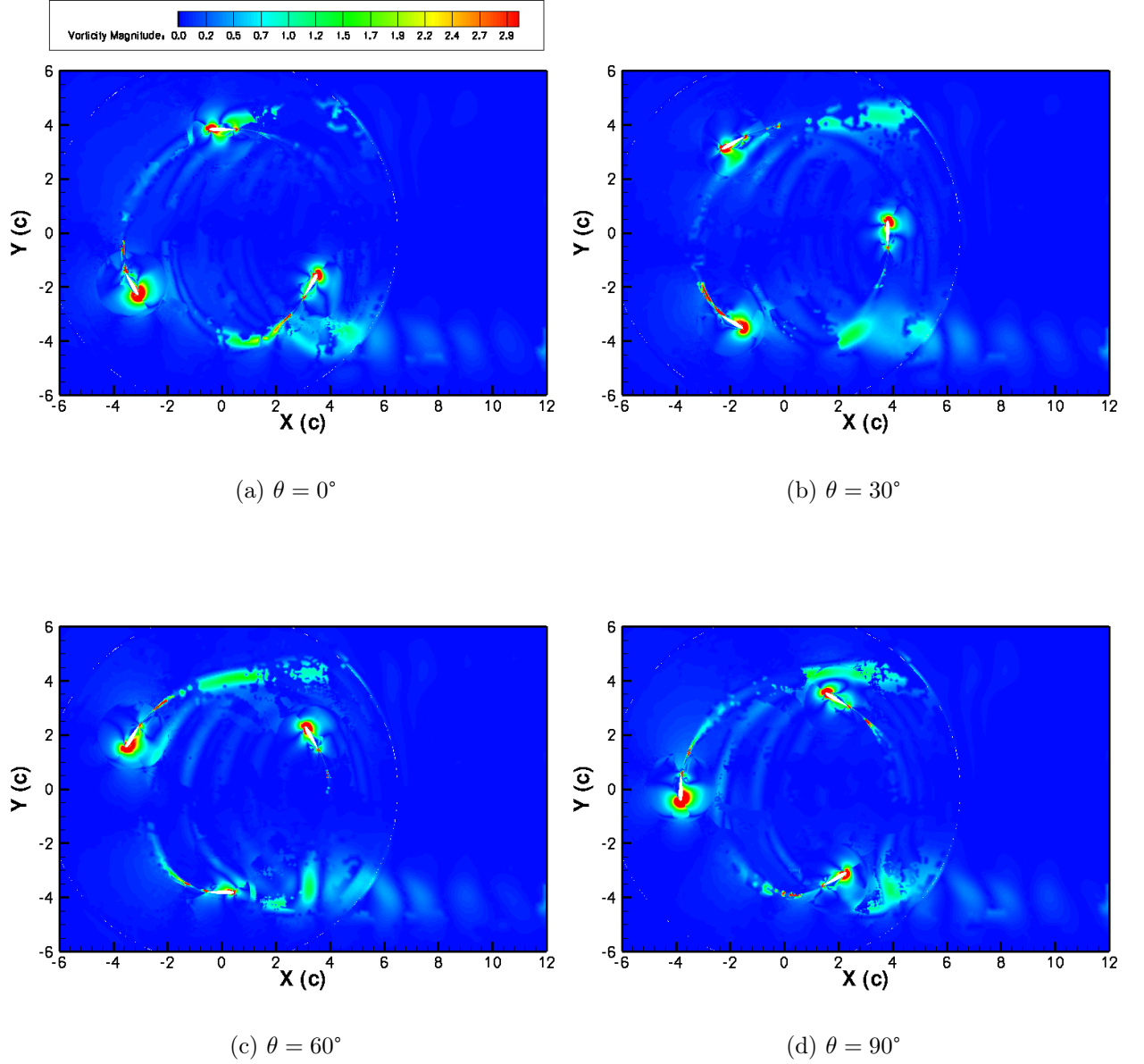


Figure 5.22: Vorticity magnitude contour plots for various azimuthal positions, θ , where $b = 50\%$ and $\lambda = 3$

In Figure 5.22 the occurrence of vortex shedding as well as the convection of the shed vortices may be observed. As one may expect, both the bound as well as the shed vorticity increase in magnitude, as the angle of attack and thus lift increase, for instance over the first quadrant from $\theta = 0^\circ$ to 90° .

5.2 Free surface modelling

Further simulations have been conducted in order to examine the effect of free surface deformation on a generic horizontal axis tidal cross-flow turbine. The importance of the free surface has been highlighted by Whelan et al. (2009) and Houlby et al. (2008) and the present study aims to further the understanding of energy extraction of a cross-flow turbine in highly blocked channel flows. The (rigid lid) simulations for $b = 12.5\%$, 25% & 50% presented in the previous section were repeated, but now a Volume of Fluid (VOF) model is used to simulate changes in the free surface associated with the energy extraction of the turbine. As discussed in Chapter 2, for the VOF simulations a User-Defined Function (UDF) is implemented, which uses a feedback loop to recursively adjust the outlet flow depth to achieve the desired inflow conditions. This iterative technique is required, because the level of energy extraction, and hence the change in depth from inlet to outlet, is not known a priori.

Firstly, we examine how the performance of the different turbines varied from the rigid lid to VOF simulations. Keeping the inflow conditions from the rigid lid the same for the VOF computations for $b = 50\%$ renders a Froude number, $Fr = 0.082$, where Fr is defined as:

$$Fr = \frac{U_\infty}{\sqrt{gh_\infty}} \quad (5.2.1)$$

and U_∞ , g and h_∞ are the free-stream velocity, gravity and upstream flow depth respectively.

With regard to the deployment of a full scale device, which is likely to be installed at a flow depth, h , of about $30 - 40$ m, $Fr = 0.082$ would result from $U_\infty = 1.41$ m/s to 1.62 m/s, which is considered a realistic flow environment. To enable a direct comparison between the VOF simulations, Fr was set to 0.082 for $b = 12.5\%$ & 25% , too. We were able to change b , but maintain both constant Fr and Re , by (i) adjusting

U_∞ , so that $Fr = 0.082$ and (ii) by adjusting μ , so that Re did not change between corresponding tests;

- for $b = 50\%$, $U_\infty = 1 \text{ m/s}$ and $\mu = 0.002575 \text{ kg/m} \cdot \text{s}$ (as for the rigid lid simulations),
- for $b = 25\%$, $U_\infty = 1.4142 \text{ m/s}$ and $\mu = 0.0036416 \text{ kg/m} \cdot \text{s}$ and
- for $b = 12.5\%$, $U_\infty = 2 \text{ m/s}$ and $\mu = 0.00515 \text{ kg/m} \cdot \text{s}$.

Figure 5.23 shows a comparison of the power and thrust curves for $b = 12.5\%$, 25% & 50% for the rigid lid (RL) and free surface (FS) simulations, where $Fr = 0.082$.

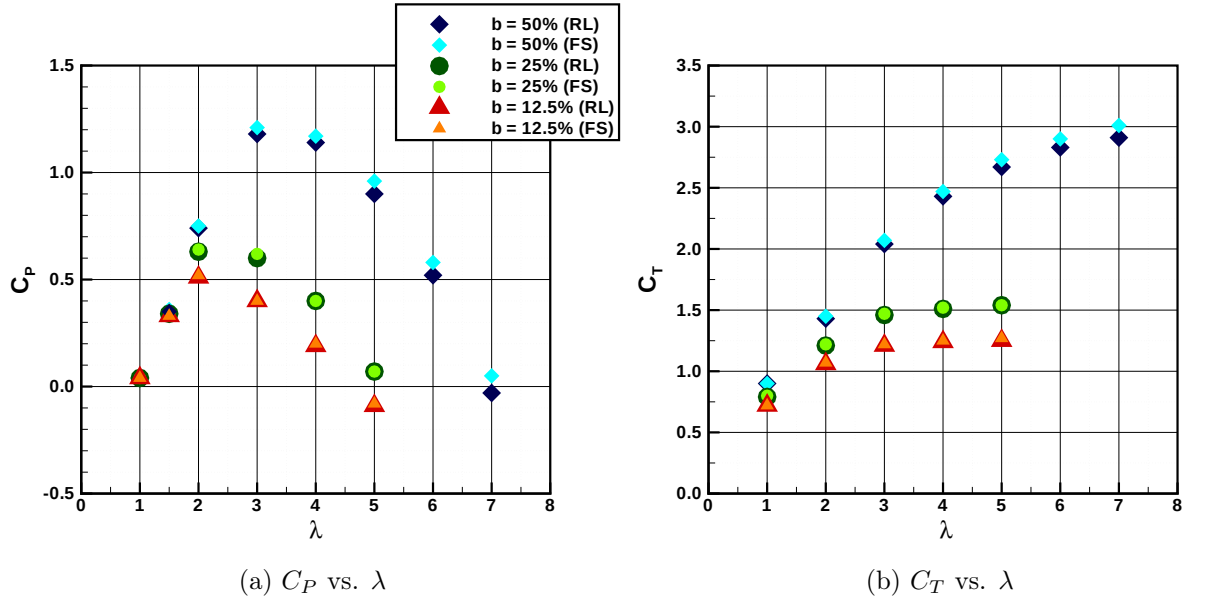


Figure 5.23: Power and thrust curves for different blockages under rigid lid (RL) and deformable free surface (FS) conditions

From Figure 5.23, it is apparent that the corresponding power and thrust curves take similar shapes for the rigid lid and VOF simulations. As discussed for the rigid lid simulations in the previous section, it may also be concluded for the VOF simulations that the higher the blockage, the greater the width of the power curve and the higher

the maximum tip speed ratio at which positive power occurs; moreover, $\max C_P$ occurs at higher λ with increasing blockage.

When comparing the corresponding results from the rigid lid and free surface simulations for C_P and C_T in Figure 5.23, it may be observed that the differences are small for $b = 12.5\%$ & 25% . However, for $b = 50\%$ at $Fr = 0.082$, a change in C_P of up to 6.7% and in C_T of up to 9.0% is noted. This increase in power and thrust can in part be attributed to an increase in the streamwise flow velocity associated with the increase in effective blockage arising from a reduction in flow depth, h , downstream of the rotor for the VOF simulations.

The close proximity of the turbine to the free surface results in a deformation of the latter. In contrast to the rigid lid simulations, where the energy is extracted in the form of pressure head, for the free surface simulations, static head (comprising potential as well as pressure head) is extracted resulting in a drop in flow depth, h . From open channel flow theory the extraction of energy from a sub-critical flow leads to a reduction in flow depth, see Massey and Ward-Smith (1998).

Figure 5.24 shows a comparison of the centreline velocity magnitude and the velocity magnitude at the cross-stream traverse at $5 D$ downstream of the turbine centre between the free surface simulation for $b = 50\%$, $\lambda = 3$ and $Fr = 0.082$ and the corresponding rigid lid computation.

From Figure 5.24a, it is apparent that the flow velocity within and downstream of the turbine is higher for the free surface simulation, as expected due to the increase in effective blockage; the increase in velocity then leads to an increase in power take-off.

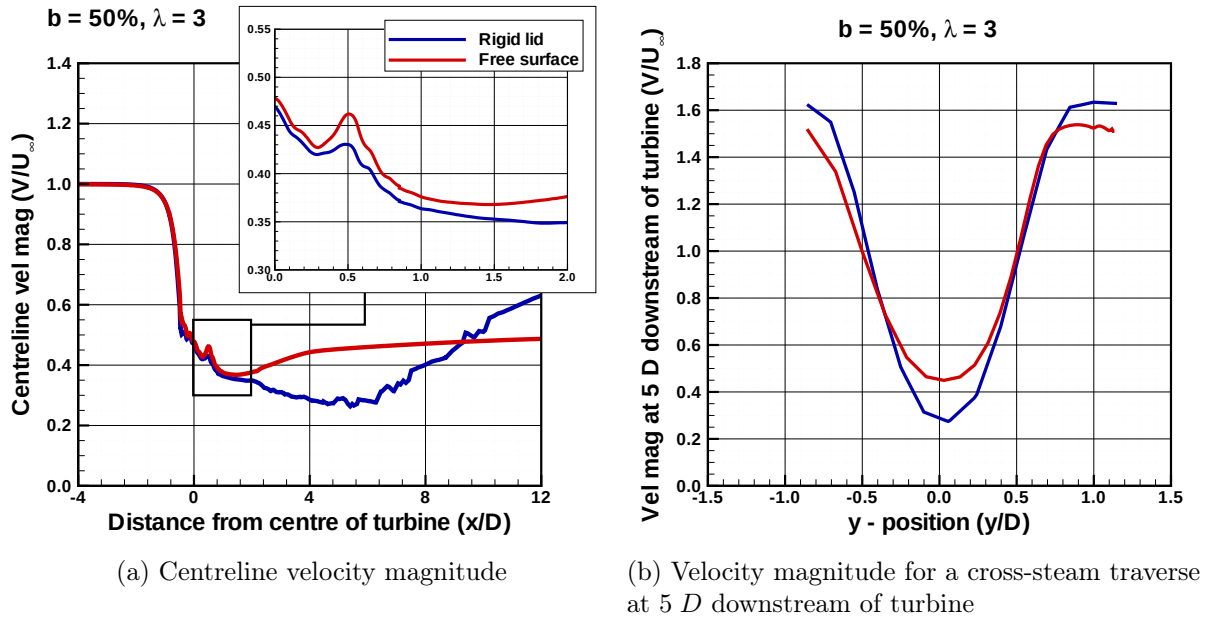


Figure 5.24: Wake velocity profiles under rigid lid and deformable free surface conditions for $b = 50\%$ & $\lambda = 3$

From Figure 5.24b, it may also be inferred that the velocity at the centreline $5 D$ downstream of the centre of the turbine is lower for the rigid lid simulations. Because the minimum velocity is significantly larger for the free surface computations, 60% at $5 D$ downstream of the turbine, the wake deficit (width of the velocity bucket) is slightly wider for the VOF simulations, as required to satisfy mass continuity.

As mentioned above, for the free surface simulations, static head is extracted resulting in a drop in flow depth, h . Figure 5.25 shows a plot of the reduction in flow depth, Δh , varying with blockage and λ for $Fr = 0.082$.

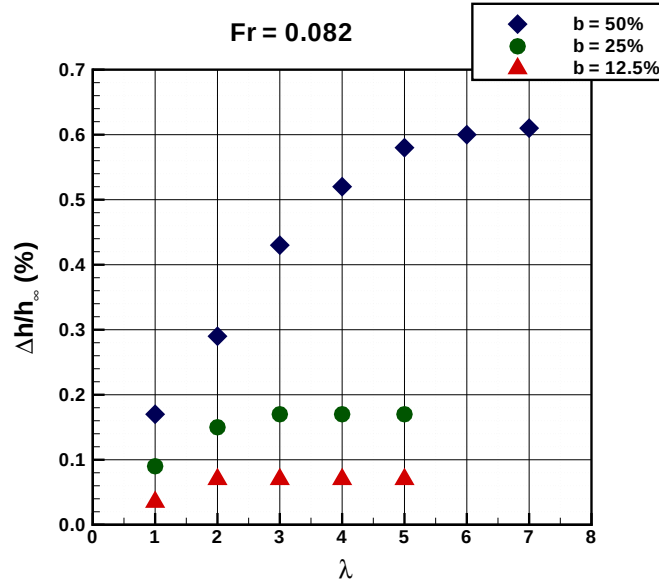


Figure 5.25: Illustration of change in flow depth for different blockages at $Fr = 0.082$

From Figure 5.25 it may be concluded that, as expected, the higher the blockage, the larger the reduction in flow depth; for instance at $\lambda = 5$;

- $\Delta h = 0.58\%$ for $b = 50\%$,
- $\Delta h = 0.17\%$ for $b = 25\%$, and
- $\Delta h = 0.07\%$ for $b = 12.5\%$.

Whilst the change in h remains constant for $b = 12.5\%$ and $b = 25\%$ for $\lambda > 2$, the change in h increases with λ for the highest blocked case. This concurs with the results for C_T shown in Figure 5.23b; for $b = 12.5\%$ and $b = 25\%$ C_T only increases marginally for $\lambda > 2$ with increasing λ , but for $b = 50\%$ significant increases in C_T are observed for an increase in λ . From Figure 5.23b & 5.25, it is apparent that Δh increases, as C_T increases.

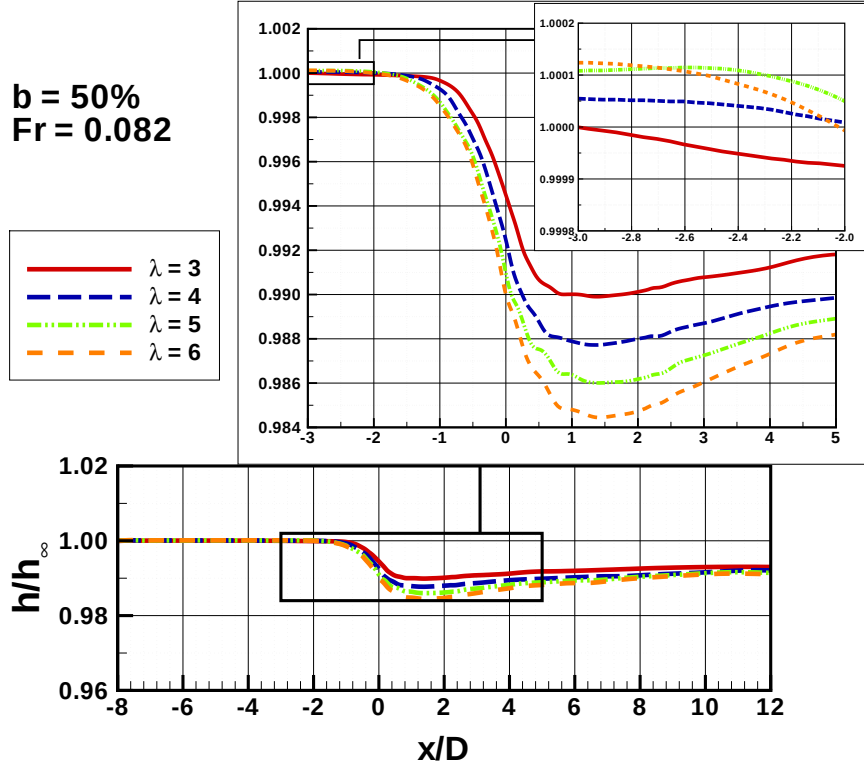


Figure 5.26: Illustration of free surface deformation due to turbine energy extraction for $b = 50\%$ & $Fr = 0.082$

Figure 5.26 shows how the flow depth, h , varies throughout the computational domain for $b = 50\%$ and $Fr = 0.082$ at $\lambda = 3, 4, 5$ & 6 . Figure 5.26 underlines that the change in flow depth across the domain increases with an increase in λ . Moreover, it is apparent that the flow depth increases relative to the inflow condition just upstream of the turbine, see at $x/D = -3$, which is due to the flow resistance presented by the turbine. Across the turbine, the flow depth drops indicating an extraction of energy. Just downstream of turbine, for instance at $1.4 D$ downstream of the turbine for $\lambda = 3$, the flow depth reaches a minimum and thereafter increases due to flow mixing until reaching h_W , the flow depth far downstream, where pressure variation may be assumed to be hydrostatic and the flow velocity uniform once again. In practice the full flow recovery and mixing takes place over a longer length scale than the computational domain and the flow at the computational domain exit is not fully remixed.

5.2.1 Basin efficiency

Due to the different mechanisms of energy extraction, it is of particular interest to compare the basin efficiencies, η , for the VOF and rigid lid simulations, where η is defined as the ratio of useful power to total power extracted from the flow field:

$$\eta = \frac{Power_{useful}}{Power_{removed}} \quad (5.2.2)$$

$$= \frac{T\omega}{\left(\int \left(P + \frac{1}{2}\rho |U|^2 + \rho gy\right) U_x dy\right)_{upstream} - \left(\int \left(P + \frac{1}{2}\rho |U|^2 + \rho gy\right) U_x dy\right)_{downstream}} \quad (5.2.3)$$

$$= \frac{T\omega}{\int_0^{h_I} (P_0 + \rho gy) U_x dy - \int_0^{h_W} (P_0 + \rho gy) U_x dy} \quad (5.2.4)$$

where h_I and h_W are respectively the flow depths far upstream at the inlet and far downstream following mixing, P the static pressure, P_0 total pressure, $|U|$ the velocity magnitude, and U_x the streamwise flow velocity component.

For both the rigid lid as well as the VOF simulations the computational domain lengths were not sufficient for the flow mixing process to be completed. Hence, it was necessary to determine the flow depth and stream energy following full flow remixing at position W downstream of the domain outlet analytically.

However, it is noted that the flow through the turbine was unaffected by not capturing the full remixing process within the computational domain. The adequacy of the domain size was confirmed by altering the distance to the far-field boundary until gross metrics, C_P and C_T , remained unaltered by further increasing the domain length.

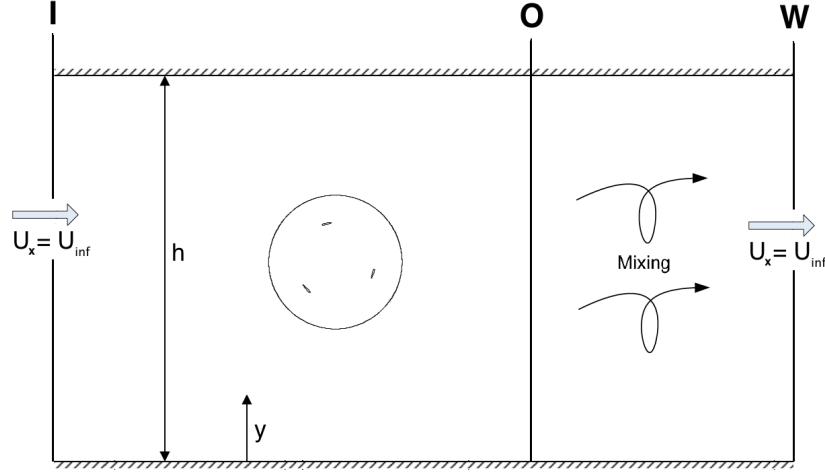


Figure 5.27: Schematic of flow mixing states for the rigid lid simulations, adapted from Houlby et al. (2008)

Figure 5.27 illustrates the flow conditions for the rigid lid simulations. Station I corresponds to the inlet of the computational domain, station O to a station between the turbine and outlet of the computational domain and W to the station far downstream at which all flow mixing has been completed. As the flow depth, h , is constant for the rigid lid case and uniform flow, i.e. $U_x = U_\infty$ & $U_y = 0$, is assumed at station W , Equation 5.2.4 may be simplified to:

$$\eta_{RL} = \frac{T\omega}{\int_0^h P_0 U_x dy \Big|_I - \int_0^h P_0 U_x dy \Big|_W} \quad (5.2.5)$$

$$\eta_{RL} = \frac{T\omega}{\int_0^h P_0 U_x dy \Big|_I - (P_W U_\infty h + \frac{1}{2} \rho U_\infty^3 h)} \quad (5.2.6)$$

where P_W is the uniform static pressure at the cross-stream traverse at station W .

Whilst the energy flux at the inlet may be computed numerically by performing the integration outlined above, $\int_0^h P_0 U_x dy$, P_W , required to determine the energy flux at station W , is unknown. Performing a linear momentum (control volume) analysis between stations O & W permits the computation of P_W :

$$\int_0^h P dy \Big|_O - P_W h = \int_0^h \rho U_x^2 dy \Big|_W - \int_0^h \rho U_x^2 dy \Big|_O \quad (5.2.7)$$

$$P_W h = \int_0^h P dy \Big|_O - \rho U_\infty^2 h + \int_0^h \rho U_x^2 dy \Big|_O \quad (5.2.8)$$

$$P_W = \frac{1}{h} \int_0^h (P + \rho U_x^2) dy \Big|_O - \rho U_\infty^2 \quad (5.2.9)$$

Hence, for the rigid lid simulations, the basin efficiency, η_{RL} , may be calculated as follows:

$$\eta_{RL} = \frac{T\omega}{\int_0^h P_0 U_x dy \Big|_I - \left[U_\infty \int_0^h (P + \rho U_x^2) dy \Big|_O - \frac{1}{2} \rho U_\infty^3 h \right]} \quad (5.2.10)$$

where the required integrals are performed numerically at stations I & O .

For the present computations, station O was taken at 6 R downstream of the turbine centre. As the energy loss in the wake is governed by momentum conservation, O can be anywhere downstream of the turbine, but should be as far upstream as possible to avoid numerical losses.

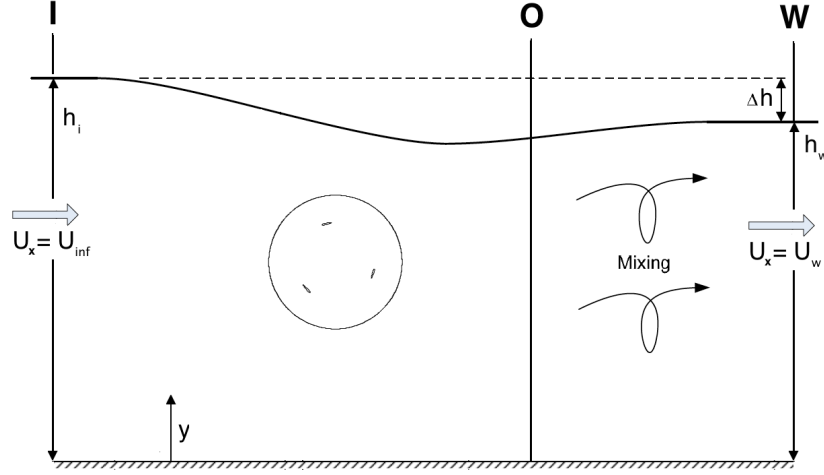


Figure 5.28: Schematic of flow mixing states for the free surface simulations, adapted from Houlsby et al. (2008)

A similar analysis is performed for the free surface simulations. Figure 5.28 illustrates the flow conditions for the VOF computations. Assuming a hydrostatic pressure variation and uniform flow, i.e. $U_x = U_W$ & $U_y = 0$, at station W and using gauge pressure, i. e. atmospheric pressure, $P_a = 0$, Equation 5.2.4 may be simplified to:

$$\eta_{FS} = \frac{T\omega}{\int_0^{h_I} (P_0 + \rho gh) U_x dy - \int_0^{h_W} (\rho gh_W + \frac{1}{2}\rho U_x^2) U_x dy} \quad (5.2.11)$$

$$= \frac{T\omega}{\int_0^{h_I} (P_0 + \rho gh) U_x dy - U_W \rho gh_W \left(h_W + \frac{U_W^2}{2g} \right)} \quad (5.2.12)$$

$$= \frac{T\omega}{\int_0^{h_I} (P_0 + \rho gh) U_x dy - \dot{m}g \left(h_W + \frac{U_W^2}{2g} \right)} \quad (5.2.13)$$

where U_W is the streamwise flow velocity component at station W and $\dot{m}$ the mass flow rate, which is defined as:

$$\dot{m} = \rho U_x h = \rho U_W h_W = \rho U_I h_I \quad (5.2.14)$$

A linear momentum (control volume) analysis is carried out between stations O & W to determine h_W :

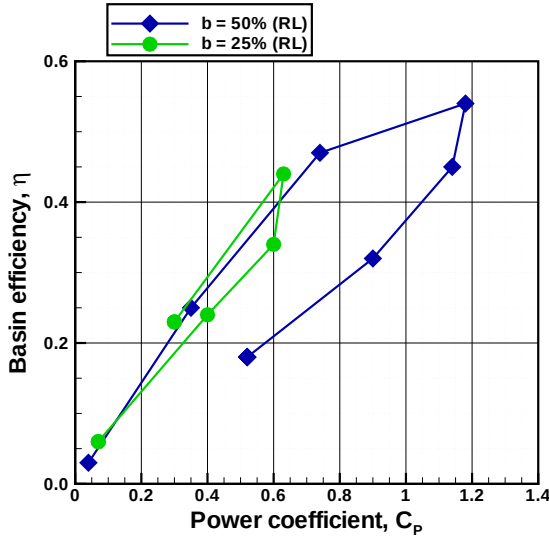
$$\int_0^{h_O} P dy \Big|_O - \int_0^{h_W} P dy \Big|_W = \int_0^{h_W} \rho U_x^2 dy \Big|_W - \int_0^{h_O} \rho U_x^2 dy \Big|_O \quad (5.2.15)$$

$$\int_0^{h_O} P dy \Big|_O - \int_0^{h_W} \rho g (h_W - y) dy \Big|_W = \rho U_W^2 h_W - \int_0^{h_O} \rho U_x^2 dy \Big|_O \quad (5.2.16)$$

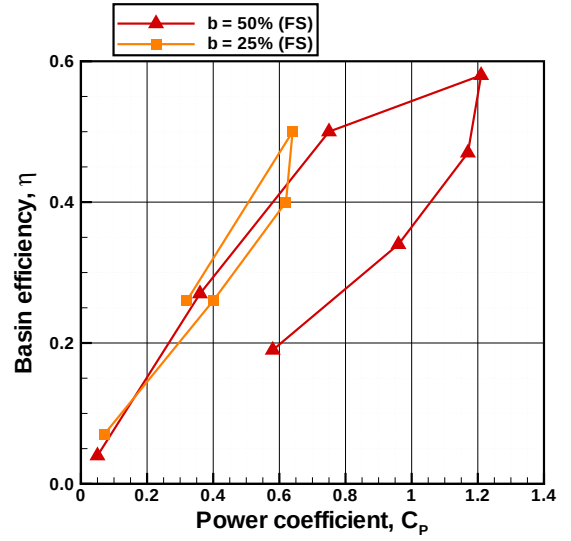
$$\Rightarrow \int_0^{h_O} (P + \rho U_x^2) dy \Big|_O = \frac{(\dot{m})^2}{\rho} \frac{1}{h_W} + \rho g \frac{h_W^2}{2} \quad (5.2.17)$$

The integral in Equation 5.2.17, $\int_0^h (P + \rho U_x^2) dy$, may be computed numerically at station O , so that Equation 5.2.17 can be solved for h_W ; U_W can then be calculated using Equation 5.2.14 and hence η_{FS} from Equation 5.2.13.

Figure 5.29a shows a comparison of the basin efficiencies computed for $b = 25\%$ & 50% using the rigid lid simulations and Figure 5.29b shows a comparison of the basin efficiencies computed for the corresponding VOF simulations at $Fr = 0.082$.



(a) Basin efficiencies for $b = 25\%$ & 50% for rigid lid simulations



(b) Basin efficiencies for $b = 25\%$ & 50% for free surface simulations

Figure 5.29: Comparison of basin efficiencies

The basin efficiency of a turbine is of great importance, as the maximum power that may be removed from a tidal basin is likely to be limited by environmental considerations, which implies that a machine with a lower η will be able to generate less useful power from a given allowable head removal from the flow field.

The maximum basin efficiency for the cross-flow turbine simulated was computed to be 0.58, observed at $b = 50\%$ & $\lambda = 3$ for the VOF simulation; at $b = 25\%$ max η was 13.8% lower than at $b = 50\%$. Generally, from Figure 5.29a & 5.29b, it is apparent that the simulations carried out for the larger blockage render higher η than the corresponding computations for the lower blockage; this is because mixing losses increase, as the difference in velocity between bypass and turbine flows increases. An increase in blockage leads to a decrease in the difference of the velocity between the bypass and turbine flows, which hence results in reduced losses and thus an increase in basin efficiency, η .

The observation that an increase in blockage not only increases the kinetic power coefficient, but also a turbine's basin efficiency is important. Given that cross-flow turbines

can present a greater effective blockage than axial-flow machines of the same diameter, they may overcome (part of) their inherently lower efficiencies relative to axial-flow turbines discussed in Section 1.3.2 by benefitting from an increased effective blockage.

As to the difference in η for the rigid lid and the corresponding VOF simulations, the shape of the η vs. C_P plots shown in Figure 5.29a & 5.29b are very similar; all curves take the shape of a horse shoe, where the “open ends” point towards the graph’s origin (0,0). However, as for C_P , the VOF simulations at $Fr = 0.082$ render slightly higher basin efficiencies, so that both VOF plots are shifted to a slightly higher range of basin efficiencies as well as C_P than the corresponding rigid lid plots.

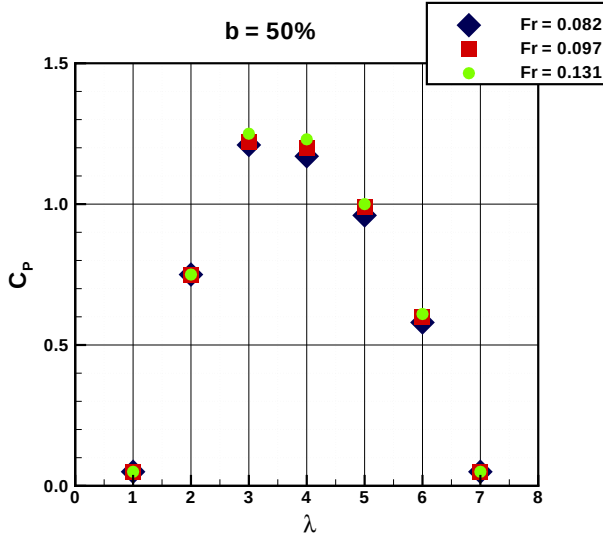
5.2.2 Froude number dependency

Furthermore, for $b = 50\%$, VOF simulations have been conducted at $Fr = 0.097$ & 0.131 to examine the effect of changes in Froude number, Fr , on turbine performance as well as free surface deformation. $Fr = 0.131$ is expected to be at the high end of the range of Fr of full scale tidal turbine flows; at a flow depth of 40 m , $Fr = 0.131$ results from a flow speed of 2.6 m/s .

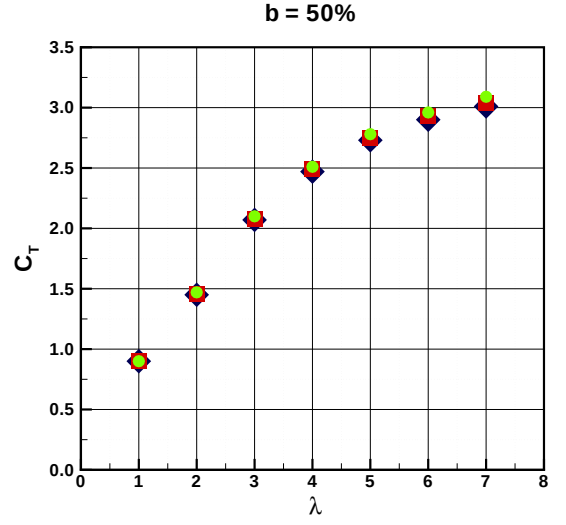
The change in Fr for constant b and Re was achieved by (i) adjusting U_∞ to attain the desired Fr and (ii) by adjusting μ , so that Re did not change between corresponding tests;

- for $Fr = 0.082$, $U_\infty = 1\text{ m/s}$ and $\mu = 0.002575\text{ kg/m} \cdot \text{s}$
- for $Fr = 0.097$, $U_\infty = 1.19\text{ m/s}$ and $\mu = 0.003064\text{ kg/m} \cdot \text{s}$
- for $Fr = 0.131$, $U_\infty = 1.6\text{ m/s}$ and $\mu = 0.00412\text{ kg/m} \cdot \text{s}$

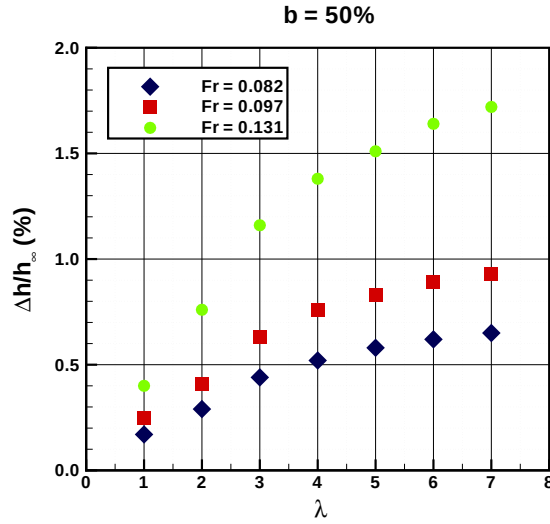
Figure 5.30 shows the power and thrust curves for $b = 50\%$ at three different Froude numbers, Fr , simulated as well as the changes in flow depth, Δh .



(a) Power curve



(b) Thrust curve



(c) Change in flow depth for $b = 50\%$ and varying Fr

Figure 5.30: Froude number dependency

From Figure 5.30a & 5.30b, it may be deduced that at low λ , an increase in Fr has little effect on both C_P & C_T , whilst at peak power and larger λ , an increase in Fr results in a small increase in both C_P & C_T ; max $C_P = 1.21$ at $\lambda = 3$ for $Fr = 0.082$, whilst max $C_P = 1.25$ at $\lambda = 3$ for $Fr = 0.131$.

Figure 5.30c shows a comparison of the changes in flow depth, Δh , for the different Fr considered. At $b = 50\%$, for all three Fr simulated, the drop in h increases with λ and

Fr ; in fact, the higher Fr , the larger Δh for an increase in λ ; max increase in drop in h from low to high λ is observed for $Fr = 0.131$, where Δh is 226.3% larger at $\lambda = 7$ than at $\lambda = 2$. Generally, it may be concluded that the reduction in h increases with an increase in Fr ; at $\lambda = 3$, which corresponds to maximum power take-off, $\Delta h = 0.44\%$ for $Fr = 0.082$, whilst $\Delta h = 1.16\%$ for $Fr = 0.131$; the largest drop in h was computed at $\lambda = 7$ for $Fr = 0.131$, where $\Delta h = 1.72\%$.

Moreover, the basin efficiencies at λ_{opt} have been compared for the three Fr simulated.

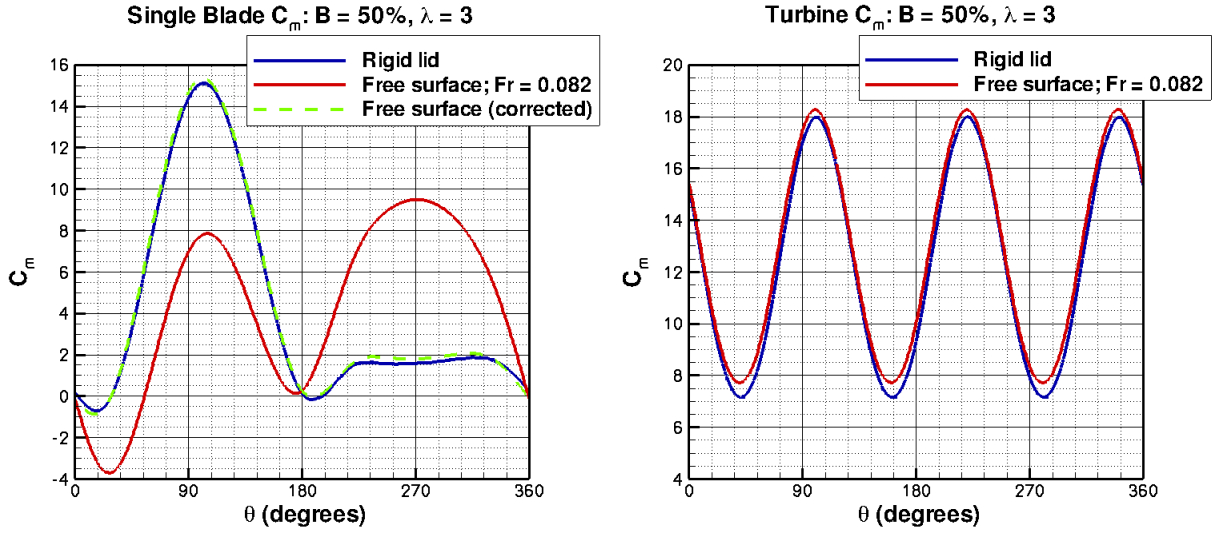
- $Fr = 0.082$, max $C_P = 1.21$, $\eta = 0.58$
- $Fr = 0.097$, max $C_P = 1.22$, $\eta = 0.58$
- $Fr = 0.131$, max $C_P = 1.25$, $\eta = 0.46$

As discussed above, the larger Fr , the higher max C_P , but whilst η is the same for $Fr = 0.082$ and $Fr = 0.097$ at λ_{opt} , η is significantly lower for $Fr = 0.131$ than for $Fr = 0.082$ & 0.097 ; i.e. an increase in Fr by 60% results in an increase in maximum power take-off of 3%, but a drop in η of 21%.

Given that the basin efficiency of a particular turbine configuration is a key indicator of the useful power a machine will produce, if the total energy removed from a tidal basin is to be limited by environmental constraints, a drop of 21% in η is of great significance.

5.2.3 Turbine and blade loads

Moreover, a significant difference between the rigid lid simulations, which were carried out assuming no gravity, and the VOF simulations, which accommodate for gravity effects, is uncovered when comparing the torque coefficient, C_m , trace of a single blade, see Figure 5.31a.



(a) C_m trace for a single blade for $b = 50\%$ and $\lambda = 3$; for the free surface case the C_m is trace also shown corrected for the effect of the hydrostatic pressure variation across the blade. (b) Turbine C_m trace for $b = 50\%$ and $\lambda = 3$

Figure 5.31: Blade and turbine torque histories for rigid lid and free surface cases

The blade torque history for the rigid lid simulation exhibits the expected shape. The azimuth angle, $\theta = 0^\circ$, corresponds to the blade's top position vertically above the centre of the turbine. As the blade reaches $\theta \approx 90^\circ$ position and thus maximum angle of incidence, the first peak in the torque trace may be observed. As to the downstream passage, it is evident that the blades contribute little to no positive torque, because they operate in perturbed flow conditions. The blade torque history for the free surface computation exhibits a very different shape, and suggests that the major positive torque contribution comes from the downstream passage.

The difference between these two traces may be explained by considering the vertical force exerted onto the blade in the VOF simulation by the hydrostatic pressure variation. It is important to note that the mass of the blades was set to zero in the simulations, whilst the blades of tidal turbines are likely to be more dense than water due to flooding.

The effect of the hydrostatic pressure variation observed across the blades in the VOF simulations was examined by subtracting C_m , equivalent to the torque coefficient around

the centre of the turbine induced by the hydrostatic pressure variation across a massless blade at $U_\infty = 0 \text{ m/s}$, from the computed torque history resulting in the corrected trace shown in Figure 5.31a.

This confirms that the difference in torque history arises from the hydrostatic pressure variation across the blade, although there remain some differences in torque history on the downstream passage which are not wholly unexpected as in this region the deformation of the free surface, and hence local flow acceleration, are largest.

This highlights how critical blade weight and buoyancy may be. With regard to generator loading and fatigue issues, the significant change in the massless blade's torque trace would be an important factor. Aside from the turbine's performance and its cost, turbine fatigue characteristics are key in identifying optimal turbine designs.

As to the (average) torque generated by a blade throughout a revolution, the torque induced by the hydrostatic pressure variation has no net effect, as its contribution cancels out over one complete cycle. Also, when comparing the turbine C_m traces, see Figure 5.31b, the contribution of the hydrostatic pressure variation across the individual blades has no effect, as its net effect cancels out across the three blades.

Figure 5.32 shows the sectional lift and drag forces experienced by a blade for $b = 50\%$ and $\lambda = 3$ from the rigid lid and free surface simulations, as it traverses a full rotation cycle. In the manner plotted, positive incidence refers to the downstream pass of the blade, whilst negative incidence refers to the upstream blade pass.

As discussed above, the hydrostatic pressure variation across an individual blade with mass $= 0 \text{ kg}$ results in an offset of the cross-stream blade force component, which affects both the lift and drag curves, as shown in Figure 5.32. When subtracting the force component due to the hydrostatic pressure variation across a blade from the computed blade forces, the lift and drag curves computed for the rigid lid and VOF simulations collapse together as anticipated from Figure 5.31a.

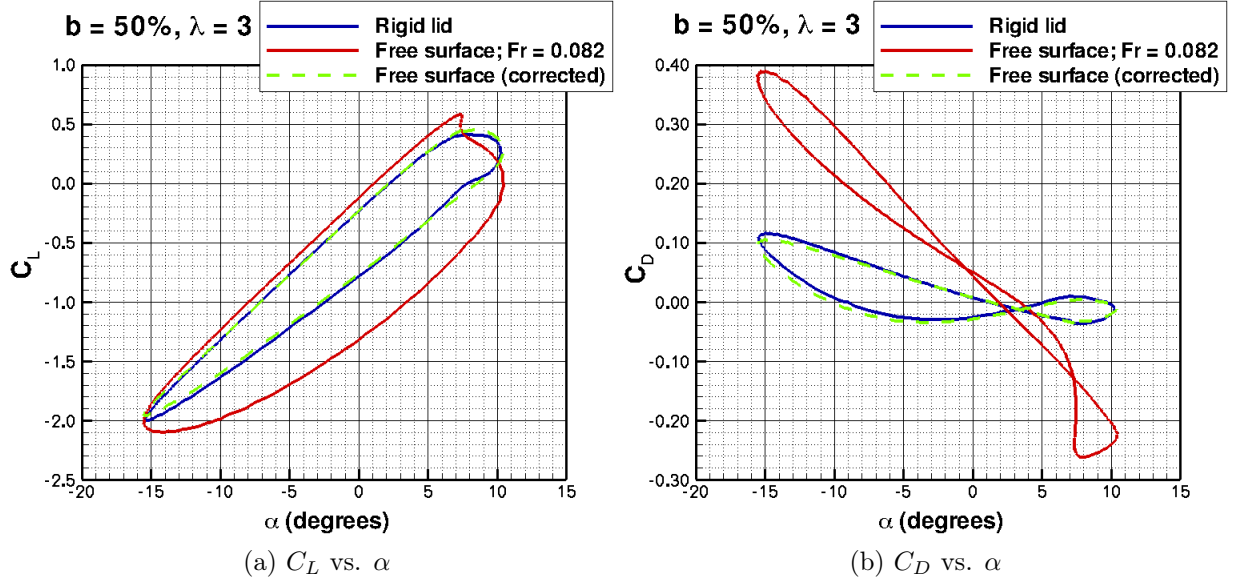


Figure 5.32: Blade coefficient histories for rigid lid and free simulations at $b = 50\%$ and $\lambda = 3$; for the free surface case the blade coefficients are also shown corrected for the effect of the hydrostatic pressure variation across the blade.

In a typical tidal basin the flow direction reverses approximately every 6 hours. One of the key advantages of horizontal axis cross-flow turbines is their multi-directionality, which means that the device's functionality/performance is independent of the direction of flow. However, the relative effect of the hydrostatic pressure difference across the blades on a blade's torque trace changes depending on the direction of flow and hence, it is of interest to examine how the blade forces as well as the turbine's performance would be affected by a reversal of flow direction.

The VOF simulations conducted at $b = 50\%$ and $Fr = 0.082$ were repeated, the flow direction was kept the same, but the orientation of rotation was reversed from anti-clockwise to clockwise; also the turbine was flipped to ensure it was spinning with blade leading edge first.

It was found that the difference in C_P , $\Delta C_P \leq 0.01$, between the corresponding clockwise and anti-clockwise rotating turbines across the entire tip speed ratio range. How-

ever, the C_m trace of a single blade changes significantly depending on the orientation of rotation, as shown in Figure 5.33.

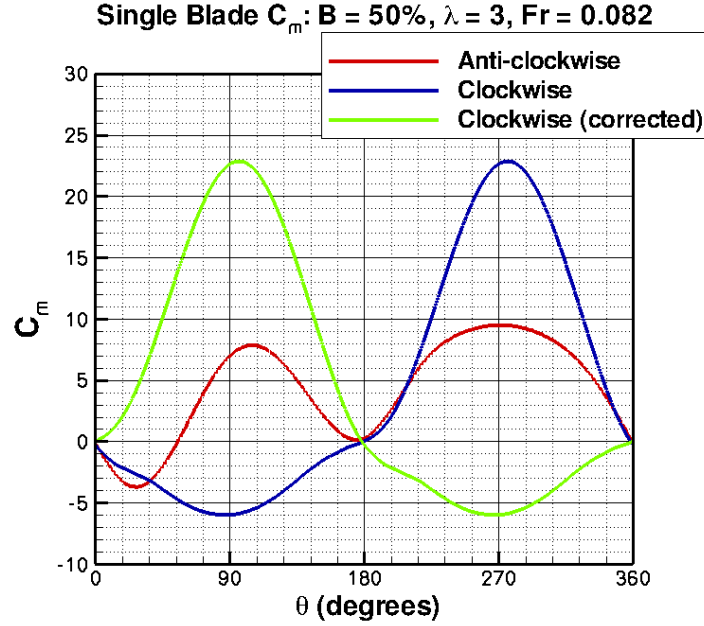


Figure 5.33: Blade torque history for $b = 50\%$ and $\lambda = 3$; for the clockwise case the C_m trace is also shown phase shifted by 180° (corrected).

Due to the change of rotational direction, quadrants 1 & 2 of the clockwise rotating turbine correspond to the downstream passage, whilst quadrants 3 & 4 correspond to the upstream passage. In order to facilitate a direct comparison between the clockwise and anti-clockwise rotating turbines, the C_m trace of a single blade of the clockwise rotating turbine, see blue trace in Figure 5.33, has been phase shifted by 180° , rendering the green trace.

It is apparent that while the hydrostatic pressure difference across the blades resulted in a more even C_m trace as to the upstream and downstream passage for the anti-clockwise rotating turbine, the differences between the up- and downstream passage observed for the rigid lid case are increased by the hydrostatic pressure variation for the clockwise rotating turbine.

As mentioned above, blade flooding may largely circumvent any potential implications

arising from the hydrostatic pressure difference across the blades. This analysis highlights that blade non-buoyancy is of importance with regard to fatigue issues.

5.3 Chapter conclusions

This chapter has explored the influence of blockage and free surface deformation on the hydrodynamics of energy extraction in a constrained channel. The effects of flow confinement were investigated by simulating flows through a three-bladed turbine with solidity 0.125 at field-test Reynolds numbers, $Re_c = O(10^6)$, for channels that resulted in cross-stream blockages of 6.25%, 12.5%, 25% and 50%. Two representations of the free surface boundary are considered; a rigid lid and a deformable free surface.

Approximating the free surface as a rigid lid, increasing the blockage was observed to lead to a substantial increase in the power coefficient; the highest power coefficient computed was 1.18. Further, the basin efficiency was found to be dependent on and increase with blockage reaching a maximum of 0.54 at the highest blockage considered. Further, the simulations for the 12.5%, 15% and 50% cross-stream blockages were repeated, but now employing a Volume of Fluid model with upstream and downstream boundary conditions, which allowed for an examination of the effect of free surface deformation on the performance of a generic horizontal axis tidal cross-flow turbine.

Comparisons between the corresponding rigid lid and free surface simulations, where Froude number, $Fr = 0.082$, rendered similar results at the lower blockages, but at the highest blockage an increase in power of up to 6.7% and an increase in basin efficiency of up to 7.4% for the free surface simulation.

For the free surface simulations with $Fr = 0.082$, the energy extraction resulted in a drop in water depth across the computational domain of between 0.05% to 0.68% depending on and increasing with both tip speed ratio and blockage.

Moreover, the effect of varying Fr was investigated. Whilst the maximum basin efficiency dropped from 0.58 to 0.46, the maximum power coefficient increased from 1.21 to 1.25 when increasing Fr from 0.082 to 0.131 and the drop in flow depth at the corresponding tip speed ratio increased from 0.44% to 1.16%.

Chapter 6

Conclusions & Future work

6.1 Conclusions

The necessity to develop an energy supply, which is clean, safe and affordable, is the fundamental driving force towards the exploitation of renewable energy sources. Towards these ends tidal stream energy generation has captured the interest of the public as well as technology developers over the last decade. Tidal stream energy is a renewable and highly predictable energy source estimated to potentially contribute up to 5% of the UK's electricity supply. However, the tidal stream turbine industry is still at an early stage in its development cycle and it has not yet identified the most cost-effective rotor design for tidal stream energy generation.

One of the turbine types focused upon by tidal stream turbine developers are cross-flow turbines. In contrast to the more conventional axial-flow designs, typically used for wind turbines, cross-flow turbines have been shown to operate with lower turbine efficiencies due to destructive interference effects of the upstream on the downstream passage. However, for a given turbine diameter cross-flow turbines have a greater theoretical potential for energy extraction than axial-flow machines, as they have a greater projected frontal area and therefore intercept a greater energy flux in the undisturbed

stream as well as present a higher effective blockage. Moreover, the design of cross-flow turbines permits the formation of single long turbine arrays, which may allow for a reduction in installation and maintenance costs.

However, the flow physics of cross-flow water turbines in confined flow conditions has not been fully understood. To this end the present study has illustrated numerical investigations of the hydrodynamic performance of generic horizontal axis marine cross-flow turbines with the objective to further the understanding of flows through such devices. The present study embodies two main investigations. The first of these is concerned with the influence of turbine solidity on turbine performance, and the second of these with the influence of blockage and free surface deformation on the hydrodynamics of energy extraction in a constrained channel.

All simulations for the present work have been conducted with the commercial CFD package ANSYS FLUENT 12.0, used as a two-dimensional, segregated, implicit, incompressible flow solver. The numerical model, using the SST $k - \omega$ turbulence model, has been validated against static, dynamic pitching blade and rotating turbine data.

Turbine solidity was investigated by simulating flows through two-, three- and four-bladed turbines, resulting in turbine solidities of 0.019, 0.029 and 0.038, respectively. The investigation was conducted for two Reynolds numbers, $Re_c = O(10^5)$ & $O(10^6)$, to reflect laboratory and field scales. Increasing the number of blades from two to four led to an increase in the maximum kinetic power coefficient from 0.43 to 0.53 for the lower Re and from 0.49 to 0.56 for the higher Re computations.

Increasing the number of blades resulted in a reduction in the streamwise flow velocity within the turbine. Consequently, the blades of the turbines with increased solidity were presented with lower angles of attack, which resulted in the entire power curve being shifted to lower tip speed ratios, as the number of blades was increased. At low tip speed ratios, power take-off is limited by stalling, so that a decrease in the angle of

attack, due to higher solidity, results in an increase in lift and hence power generated, whilst at high tip speed ratios, low angles of attack are the limiting factor, so that a decrease in the angle of attack due to higher solidity results in lower lift and thus power.

Also, it was observed that dynamic stall occurred at the lowest tip speed ratios for the lower Re simulations on both the upstream and downstream blade passes. However, the net effect of dynamic stall on turbine performance was found to be negative for the turbine configuration investigated.

In addition to an increase in maximum power, increasing Re was found to result in a widening and shift of the power curve to a higher range of tip speed ratios.

The effects of flow confinement were investigated by simulating flows through a three-bladed turbine with a turbine solidity of 0.125 at field-test Reynolds numbers, $Re_c = O(10^6)$, for channels that resulted in cross-stream blockages, b , from 6.25% to 50%. Two representations of the free surface boundary are considered; a rigid lid and a deformable free surface.

Approximating the free surface as a rigid lid, increasing the blockage was observed to lead to a substantial increase in the power coefficient; the highest power coefficient at $b = 6.25\%$ computed was 0.45 and at $b = 50\%$ was 1.18. The present work has identified the fluid mechanism by which actual turbine blades may extract increased power through higher localised flow velocities and greater angles of attack, when presented with a blocked flow. Moreover, it was determined that increasing blockage resulted in higher streamwise flow velocities through the turbine, that increased the width of the power curve and the maximum tip speed ratio at which positive power occurs; also, max power occurs at a higher tip speed ratio with increasing blockage.

Further, the simulations for the 12.5%, 15% and 50% cross-stream blockages were repeated, but now employing a Volume of Fluid model with upstream and downstream

boundary conditions, which allowed for an examination of the effect of free surface deformation on the performance of a generic horizontal axis tidal cross-flow turbine. Direct comparison between rigid lid and deformable free surface simulations have hitherto not been conducted in the literature.

Comparisons between the corresponding rigid lid and free surface simulations, where Froude number, $Fr = 0.082$, rendered similar results at the lower blockages, but at the highest blockage an increase in power of up to 6.7% for the free surface simulation over that achieved with a rigid lid boundary condition.

For the free surface simulations with $Fr = 0.082$, the energy extraction resulted in a drop in water depth across the computational domain of between 0.05% and 0.68% depending on and increasing with both tip speed ratio and blockage.

Moreover, the effect of changing Fr from 0.082 to 0.097 and 0.131 was investigated. The maximum power coefficient increased from 1.21 to 1.25 when increasing Fr from 0.082 to 0.131 and the drop in flow depth at the corresponding tip speed ratio increased from 0.44% to 1.16%.

Furthermore, the present study compared the basin efficiency, defined as the ratio of useful power to total power extracted from the flow, of various turbine configurations simulated. The basin efficiency of a turbine is of great importance, as the maximum power that may be extracted from a tidal basin is likely to be limited by environmental constraints. The maximum basin efficiency computed was 0.58, which occurred for the free surface simulation at $Fr = 0.082$, $b = 50\%$ and tip speed ratio of 3. Increasing Fr to 0.131 resulted in a lower maximum basin efficiency of 0.46. The rigid lid simulations rendered a maximum basin efficiency of 0.54, which is 6.9% lower than that for the corresponding free surface simulation at $Fr = 0.082$. Moreover, the maximum basin efficiency computed for $Fr = 0.082$ and $b = 25\%$ was 0.50, which is 13.8% lower than for the corresponding simulation at $b = 50\%$. These results show that an increase in the effective blockage not only increases the power coefficient, but also a turbine's

basin efficiency. Given that cross-flow turbines can present a greater effective blockage than axial-flow machines, they may overcome (part of) their inherently lower turbine efficiencies arising from destructive interference effects by benefitting from an increased effective blockage.

6.2 Contribution of thesis

The main contributions of this thesis are the following:

- In-depth study of the effect of turbine solidity on turbine performance:

As discussed in Chapter 1, the effect of solidity on turbine performance has been examined before, but primarily experimentally or with lower order models. In this thesis, the effect of varying the number of blades on the flow physics of a cross-flow turbine has been studied in depth for the first time. The novelty rests in examining how and why the individual blades of a turbine configuration generate less (or more) power than the individual blades of a turbine with a different solidity.

- In-depth study of the effect of blockage on turbine performance:

Flow confinement effects have been identified as one of the key differences between wind and tidal energy generation. In this thesis, we have examined the effect of varying blockage on the flow physics of cross-flow turbines. The present work has identified the fluid mechanism by which actual turbine blades may extract increased power through higher localised flow velocities and greater angles of attack, when presented with a blocked flow.

- Comparison of corresponding simulations employing rigid lid and free surface boundary conditions:

For the first time, the results of cross-flow turbine CFD simulations employing a free-surface boundary condition have been presented. In the present work, we have identified how the energy is extracted and (locally) what effect the free surface deformation has. Moreover, for this thesis comparisons of simulations of the same (cross-flow) turbine configuration using (i) rigid lid and (ii) free-surface boundary conditions have been

carried for the first time. Future work will need to show, whether the results presented in this thesis as to the comparison of rigid lid and free-surface simulations may even be rotor independent.

- Introduction and comparison of basin efficiencies of (marine) cross-flow turbines:

As discussed above, the basin efficiency of a turbine will be an important criterion as to choosing the optimum rotor design for maximum tidal energy generation. In this thesis, we have computed the basin efficiency of a marine cross-flow turbine for the first time and examined how it is affected by changes in Froude number and flow blockage.

6.3 Future work

There are a number of different areas, which the present author would like to propose as an extension of the current study.

The structural integrity of the various cross-flow turbines simulated and tested hydrodynamically needs to be examined. The structural integrity, which is affected by changes in the design parameters, will have a feedback effect on the (optimum) hydrodynamic design. For instance, increasing the effective blockage has been shown by the present study to increase the performance of cross-flow turbines, both in terms of the kinetic power coefficient as well as basin efficiency. Also, increasing the turbine solidity has been shown to potentially result in a higher power take-off. However, as discussed in Chapter 4 and 5, both increasing the number of blades as well as the effective blockage leads to an increase in the turbine's thrust, which will result in higher stress loadings. An extensive stress analysis will need to show whether design optimisations derived from hydrodynamic performance tests, as in the present study, are implementable.

This leads onto the investigation of design parameters, which have not been studied for the present work. For instance, cross-flow turbines, particularly if arranged in arrays, are likely to require thick blade sections and it would be interesting to examine what effect changes in blade thickness have on the hydrodynamics of cross-flow turbines. Also, it would be important to investigate whether and by how much a turbine's performance could be improved when off-setting the blades by a fixed pitch angle and what the optimum fixed pitch angle would be.

Moreover, in order to evaluate the feasibility of a horizontal axis cross-flow turbine for tidal energy extraction, the effects of yawed flows need to be studied. This would require three-dimensional (3D) simulations. Also, 3D computations would be required to investigate the performance of cross-flow turbines in turbine farm arrangements. The results from the present study indicate that increases in the effective blockage

can positively influence the performance of a cross-flow turbine and it remains to be scrutinised how the performance of cross-flow turbine arrays would compare to that of axial-flow turbine farms. Moreover, the maximum drop in flow depth, particularly for high blockages, is important to simulate, as it will affect the feasibility of particular cross-flow turbine arrangements.

Furthermore, the effects of free surface waves need to be studied and what the optimum position of a turbine in the water column would be. For instance, in EC (1996) it was suggested to avoid the top 8 *m* due to surface wave effects, but this requires further investigation.

Appendix

Journal papers

Consul, C.A., Willden, R.H.J. and McIntosh, S.C. (2012). An investigation of the influence of free surface effects on the hydrodynamic performance of marine cross-flow turbines. *Philosophical Transactions of the Royal Society A*. (to appear in)

Conference papers

Consul, C.A., Willden, R.H.J. and McIntosh, S.C. (2011). An investigation of the influence of free surface effects on the hydrodynamic performance of marine cross-flow turbines. *9th European Wave and Tidal Energy Conference*, Southampton, UK.

Consul, C.A. and Willden, R.H.J. (2010). Influence of flow confinement on the performance of a cross-flow turbine. *3rd International Conference on Ocean Energy*, Bilbao, Spain.

Consul, C.A., Willden, R.H.J., Ferrer, E. and McCulloch, M.D. (2009). Influence of solidity on the performance of a cross-flow turbine. *8th European Wave and Tidal Energy Conference*, Uppsala, Sweden.

Bibliography

- Abbott, I. and von Doenhoff, A. (1959). *The Theory of Wing Sections*. Dover Publications.
- Anderson, J. (1985). *Computational Fluid Dynamics: The basics with applications*. McGraw Hill.
- ANSYS Inc., . (2006). *ANSYS CFX Manuel*.
- ANSYS Inc., . (2009). *ANSYS FLUENT 12.0 Manuel*.
- Antheaume, S., Maitre, T., and Achard, J.-L. (2008). Hydraulic Darrieus turbines efficiency for free fluid flow conditions versus power farm conditions. *Renewable Energy*, 33:2186 – 2198.
- Atlantis Resources Corporation, . (2011). Last accessed on 01/10/2011. <http://www.atlantisresourcescorporation.com/>.
- Barton, J. T. and Pulliam, T. H. (1986). Airfoil computation at high angles of attack, invscid and viscous phenomena. *AIAA journal*, 24:705 – 712.
- Belloni, C. and Willden, R. (2011). Flow field and performance analysis of bidirectional and open-centre ducted turbines. In *9th European Wave and Tidal Energy Conference*, Southampton, UK.
- BERR (2008). Atlas of UK marine renewable energy sources. Technical report, Department for Business Enterprise & Regulatory Reform.

- Betz, A. (1920). Das Maximum der theoretisch moeglichen Ausnutzung des Windes durch Windmotoren. *Zeitschrift fuer das gesamte Turbinenwesen*, 26:307 – 309.
- Bhaskaran, R. and Collins, L. (2011). Introduction to CFD basics. dragonfly.tam.cornell.edu/teaching/mae5230-cfd-intro-notes.pdf.
- Black & Veatch, . (2005). Tidal stream energy resource and technology summary report. Technical report, for the Carbon Trust.
- Blunden, L. and Bahaj, A. (2006). Initial evaluation of tidal stream energy resources at Portland Bill, UK. *Journal of Renewable Energy*, 31:121 – 132.
- Boussinesq, J. (1877). Essai sur la theorie des eaux courantes. *Memoires presentes par divers savants a l'Academie des Sciences, Paris, France*, 23:1–690.
- Brochier, G., Fraunie, P., Beguier, C., and Paraschivoiu, I. (1986). Water channel experiments of dynamic stall on Darrieus wind turbine blades. *Journal of Propulsion and Power*, 2:445–449.
- Bryden, I. G., Couch, S. J., Owen, A., and Melville, G. (2007). Tidal current resource assessment. *Proceedings of the Institution of Mechanical Engineers, Part A: Journal of Power and Energy*, 221:125–135.
- Burton, T., Sharpe, D., Jenkins, N., and Bossanyi, E. (2001). *Wind Energy Handbook*. John Wiley & Sons, Ltd.
- Carr, L. W. (1988). Progress in analysis and prediction of dynamic stall. *Journal of Aircraft*, 25:6–17.
- Clarke, J., Connor, G., Grant, A., Johnstone, C., and Ordonez-Sanchez, S. (2008). Contra-rotating marine current turbines: Performance in field trials and power train developments. In *10th World Renewable Energy Congress*, Glasgow, UK.

- Clean Current, . (2011). Last accessed on 01/10/2011. <http://www.cleancurrent.com/>.
- Coiro, D., Nicolosi, F., De Marco, A., Melone, S., and Montella, F. (2005). Flow curvature effects on dynamic behaviour of a novel vertical axis tidal current turbine: Numerical and experimental analysis. In *24th International Conference on Offshore Mechanics and Arctic Engineering*, Halkidiki, Greece.
- Consul, C. (2008). Hydrodynamic analysis and design optimisation of a marine tidal cross-flow turbine. Technical report, University of Oxford.
- Dai, Y. M. and Lam, W.-H. (2009). Numerical study of straight-bladed Darrieus-type tidal turbines. *Proceedings of Institution of Civil Engineers: Energy*, 162:67 – 76.
- Darrieus, G. (1931). Turbine having its rotating shaft transverse to the flow of the current, US patent 1835018.
- DECC (2011). Digest of United Kingdom energy statistics. Technical report, Department of Energy and Climate Change.
- Dillon, L. J. and Woolf, D. K. (2008). An initial evaluation of potential tidal stream development sites in Pentland Firth, Scotland. In *10th World Renewable Energy Congress*, Glasgow, UK.
- Doerner, H. (1975). Efficiency and economic comparison of different WEC - (wind energy converter) rotor systems. *Appropriate technologies in semiarid areas: wind and solar energy for water supply; German foundation for international development*, pages 43 – 70.
- Draper, S., Houlsby, G., Oldfield, M., and Borthwick, A. (2010). Modelling tidal energy extraction in a depth-averaged coastal domain. *IET Renewable Power Generation*, 4(6):545 – 554.

- DTI (2001). The commercial prospects for tidal stream power. Technical report, The Department of Trade and Industry.
- DTI (2003). Energy white paper - our energy future - creating a low carbon economy. Technical report, The Department of Trade and Industry.
- Durbin, P. (1995). Separated flow computations with the k-epsilon-v-squared model. *AIAA Journal*, 33:659 – 664.
- Earthsky (2011). Last accessed on 01/10/2011. <http://earthsky.org/space/whats-true-and-false-about-the-march-19-supermoon>.
- EC (1996). The exploitation of tidal marine currents. Technical report, European Commission, Report EUR16683EN.
- Faure, T. (1984). Experimental results of a Darrieus type vertical axis rotor in a water current. Technical report, National Research Council of Canada.
- Faure, T., Pratte, B., and Swan, D. (1986). Darrieus hydraulic turbine - model and field experiments. *American Society of Mechanical Engineers, Fluids Engineering Division*, 43:123 – 129.
- Ferziger, J. and Peric, M. (2002). *Computational methods for fluid dynamics*. Springer-Verlag, 3rd edition.
- FLUENT Inc., . (2003). *GAMBIT 2.1 Manuel*.
- Foreman, K. (1981). Preliminary design and economic investigations of diffuser augmented wind turbines (DAWT). Technical report, Solar Energy Research Institute, U.S.
- Fraenkel, P. and Musgrove, P. (1979). Tidal and river current energy systems. In *International conference on future energy concept*, volume 17, pages 114 – 117. IEE.

- Fujisawa, N. and Shibuya, S. (2001). Observations of dynamic stall on Darrieus wind turbine blades. *Journal of Wind Engineering and Industrial Aerodynamics*, 89(2):201–214.
- Gaden, D. (2007). *An investigation of river kinetic turbines*. PhD thesis, University of Manitoba, Canada.
- Gant, S. and Stallard, T. (2008). Modelling a tidal turbine in unsteady flow. In *8th International Offshore and Polar Engineering Conference*, Vancouver, Canada.
- Garrett, C. and Cummins, P. (2005). The power potential of tidal currents in channels. *Proceedings of the Royal Society A: Mathematical, Physical and Engineering Sciences*, 461(2060):2563 – 2572.
- GCK Technology, . (2011). Last accessed on 01/10/2011. <http://www.gcktechnology.com/GCK/pg2.html>.
- Gorlov, A. M. (1998). Extracting energy from ocean waves and currents using the helical turbine. In *17th International Conference on Offshore Mechanics and Arctic Engineering*, Lisbon, Portugal.
- Gorlov, A. M. (2003). The helical turbine and its applications for tidal and wave power. In *OCEANS Conference*, San Diego, U.S.
- Gregorek, G. (1983). Developments in Sandia laminar airfoils for wind turbine application. Technical report, Sandia National Laboratories, U.S.
- Gretton, G. and Bruce, T. (2005). Preliminary results from analytical and numerical models of a variable-pitch vertical-axis tidal current turbine. In *6th European Wave and Tidal Energy Conference*, Glasgow, Scotland.
- Gretton, G., Bruce, T., and Ingram, D. (2009). Hydrodynamic modelling of a verti-

- cal axis tidal current turbine using CFD. In *8th European Wave and Tidal Energy Conference*, Uppsala, Sweden.
- Hammerfest Strom, . (2011). Last accessed on 01/10/2011. <http://www.hammerfeststrom.com/products/>.
- Houlsby, G., Draper, S., and Oldfield, M. (2008). Application of linear momentum actuator disc theory to open channel flow. Technical report, University of Oxford.
- Hwang, I. S., Lee, Y. H., and Kim, S. J. (2009). Optimization of cycloidal water turbine and the performance improvement by individual blade control. *Applied Energy*, 86:1532 – 1540.
- Kaplan, W. (1991). *Advanced Calculus*. Addison Wesley Publishing Company, 4th edition.
- Kawase, M. and Thyng, K. (2010). Three-dimensional hydrodynamic modelling of inland marine waters of Washington State, United States, for tidal resource and environmental impact assessment. *IET Renewable Power Generation*, 4(6):568 – 578.
- Kirke, B. (1998). *Evaluation of self-starting vertical axis wind turbines for stand-alone applications*. PhD thesis, Griffith University, Australia.
- Kirke, B. (2005). Developments in ducted water current turbines. Technical report, Cyberiad, Australia.
- Kirke, B. and Lazauskas, L. (1991). Enhancing the performance of a vertical axis wind turbine using a simple variable pitch systems. *Wind Engineering*, 15, No. 4:187 – 195.
- Klaptocz, V., Rawlings, G., Nabavi, Y., Alidadi, M., Y., L., and Calisal, S. (2007).

- Numerical and experimental investigation of a ducted vertical axis tidal current turbine. In *7th European Wave and Tidal Energy Conference*, Porto, Portugal.
- Laneville, A. and Vittecoq, P. (1986). Dynamic stall: The case of the vertical axis wind turbine. *Journal of Solar Energy Engineering*, 108(2):140–145.
- Langtry, R. B. and Menter, F. R. (2009). Correlation-based transition modeling for unstructured parallelized computational fluid dynamics codes. *AIAA Journal*, 47(12):2894 – 2906.
- Launder, B. (1975). Progress in the development of a Reynolds-stress turbulence closure. *Journal of Fluid Mechanics*, 68(3):537 – 566.
- Lunar Energy, . (2008). Personal discussion at All-Energy, Aberdeen, Scotland.
- Lunar Energy, . (2011). Last accessed on 01/10/2011. <http://www.lunarenergy.co.uk/>.
- Marine Current Turbines, . (2011). Last accessed on 01/10/2011. <http://www.marineturbines.com/>.
- Massey, B. S. and Ward-Smith, J. (1998). *Mechanics of fluids*. Stanley Thornes, 7th edition.
- Mayda, E. A. and van Dam, C. P. (2002). Bubble-induced unsteadiness on a wind turbine airfoil. *Transactions of the ASME. Journal of Solar Energy Engineering*, 124(4):335–44.
- McAdam, R. (2007). Design, build and test a horizontal axis marine turbine. Technical report, University of Oxford.
- McAdam, R. (2008). The design, test and analysis of the Transverse Horizontal Axis Water Turbine. Technical report, University of Oxford.

- McAdam, R., Houlsby, G., Oldfield, M., and McCulloch, M. (2010). Experimental testing of the Transverse Horizontal Axis Water Turbine. *IET Renewable Power Generation*, 4(6):510 – 518.
- McCroskey, W. (1981). The phenomenon of dynamic stall. Technical report, NASA Technical Memorandum 81624.
- McIntosh, S., Fleming, C., and Willden, R. (2010). WG3 WP1 D1: Report on model setup for horizontal axis axial flow turbines. Technical report, Energy Technologies Institute, University of Oxford.
- Menter, F. (1994). Two-equation eddy-viscosity turbulence models for engineering applications. *AIAA Journal*, 32(8):1598 – 1605.
- Merchant, I. (2008). The two towers. In *All-Energy*, Aberdeen, Scotland.
- Mertens, S., van Kuik, G., and van Bussel, G. (2003). Performance of an H-Darrieus in the skewed flow on a roof. *Journal of Solar Energy and Engineering*, 125:433–440.
- Migliore, P. G. (1983). Comparison of NACA 6-series and 4-digit airfoils for Darrieus wind turbines. *Journal of Energy*, 7(4):291–292.
- Migliore, P. G., Wolfe, W. P., and Fanucci, J. B. (1980). Flow curvature effects on Darrieus turbine blade aerodynamics. *Journal of Energy*, 4(2):49–55.
- Myers, L. and Bahaj, A. (2005). Simulated electrical power potential harnessed by marine current turbine arrays in the Alderney Race. *Journal of Renewable Energy*, 30:1713 – 1731.
- Myers, L. and Bahaj, A. (2006). Wake studies of a 1/30th scale horizontal axis marine current turbine. *Ocean Engineering*, 34:758 – 762.
- Newman, B. G. (1983). Actuator-disc theory for vertical-axis wind turbines. *Journal of Wind Engineering and Industrial Aerodynamics*, 15(1-3):347–355.

- Nicholls-Lee, R. F. and Turnock, S. R. (2008). Tidal energy extraction: Renewable, sustainable and predictable. *Science Progress*, 91(Part 1):81–111.
- OpenHydro (2011). Last accessed on 01/10/2011. <http://www.openhydro.com/home.html>.
- Osalusi, E., Side, J., and Harris, R. (2009). Structure of turbulent flow in EMEC’s tidal energy site. *International Communications in Heat and Mass Transfer*, 36:422 – 431.
- Paraschivoiu, I. (1983). Predicted and experimental aerodynamic forces on the Darrieus rotor. *Journal of energy*, 7(6):610 – 615.
- Paraschivoiu, I. (2002). *Wind turbine design: with emphasis on Darrieus concept*. Polytechnic International Press.
- Piziali, R. (1994). 2-D and 3-D oscillating wing aerodynamics for a range of angles of attack including stall. Technical report, NASA Technical Memorandum 4632.
- Ponte di Archimede International, . (2011). Last accessed on 01/10/2011. http://www.pontediarchimede.it/language_us/progetti_det.mvd?RECID=2&CAT=002&SUBCAT=&MODULO=Progetti_ENG&returnpages=&page_pd=d.
- Potter, M. and Wiggert, D. (2002). *Mechanics of fluids*. Brooks/Cole.
- Pugh, D. (1996). *Tides, surges and mean sea-level*. John Wiley & Sons, Ltd.
- Pulliam, T. and Vastano, J. (1990). Chaotic flow over an airfoil. *Lecture Notes in Physics*, (371):106 – 106.
- Quiet Revolution, . (2011). Last accessed on 01/10/2011. <http://www.quietrevolution.com/>.
- Salter, S. H. and Taylor, J. R. M. (2006). Vertical-axis tidal-current generators and the Pentland Firth. *Power and Energy*, 221:181 – 199.

- Savage, A. (2007). Tidal power in the UK - tidal technologies overview. Technical report, Sustainable Development Commission.
- Sharpe, D. (1990). Wind turbine aerodynamics. In *Wind Energy Conversion Systems*. Prentice Hall.
- Sheldahl, R. and Klimas, P. (1981). Aerodynamic characteristics of seven symmetrical airfoil sections through 180-degree angle of attack for use in aerodynamic analysis of vertical axis wind turbines. Technical report, Sandia National Laboratories, U.S.
- Shi, Y. (2008). PIV for THAWT. Technical report, University of Oxford.
- Shiono, M., Suzuki, K., and Kiho, S. (2000). Experimental study of the characteristics of a Darrieus turbine for tidal power generation. *Electrical Engineering in Japan*, 132(3):38–47.
- Spalart, P. and Allmaras, S. (1992). A one-equation turbulence model for aerodynamic flows. *AIAA-Paper 92-0439*.
- Strickland, J. (1975). The Darrieus turbine: a performance prediction model using multiple streamtubes. Technical report, SAND75-0431, Sandia National Laboratories, U.S.
- Strickland, J., Webster, B., and T., N. (1981). Vortex model of the Darrieus turbine: An analytical and experimental study. Technical report, SAND81-7017, Sandia National Laboratories, U.S.
- Swan, D. and Farrell, J. (1983). Field testing of a 2.4m diameter vertical axis watermill. Technical report, National Research Council of Canada.
- Swanturbines, . (2011). Last accessed on 01/10/2011. <http://www.swanturbines.co.uk/home.htm>.

- Takamatsu, Y., Furukawa, A., and Okuma, K. (1985). Study on hydrodynamic performance of Darrieus-type cross-flow water turbines. *Bulletin of The Japanese Society of Mechanical Engineers*, 28, No. 240.
- Tang, D. M. and Dowell, E. H. (1995). Experimental investigation of three-dimensional dynamic stall model oscillating in pitch. *Journal of Aircraft*, 32(5):1062–1071.
- TidalStream (2011). Last accessed on 01/10/2011. <http://www.tidalstream.co.uk/>.
- Tucker, P. G. (2006). Turbulence modelling of problem aerospace flows. *International Journal for Numerical Methods in Fluids*, 51(3):261–283.
- Turby, . (2011). Last accessed on 01/10/2011. <http://www.turby.nl/>.
- Uglow, G. (2008). Modelling of a horizontal axis marine turbine using Computational Fluid Dynamics. Technical report, University of Oxford.
- van Bussel, G. (2007). The science of making more torque from wind: Diffuser experiments and theory revisited. *Journal of Physics, Conference Series* 75.
- Vaz, G., Van Der Wal, R., Mabilat, C., and Gallagher, P. (2007). Viscous flow computations on smooth cylinders. A detailed numerical study with validation. In *26th International Conference on Offshore Mechanics and Arctic Engineering*, San Diego, U.S.
- Versteeg, H. and Malalasekera, W. (2007). *An introduction to Computational Fluid Dynamics: The Finite Volume Method*. Pearson Education Ltd., 2nd edition.
- Vittecoq, P. and Laneville, A. (1982). Etude en soufflerie d’un rotor de type Darrieus. Technical report, Sherbrooke University, Report: MEC-82.
- Weick, F. E. (1926). Propeller design: Practical application of the blade element theory. Technical report, United States National Advisory Committee for Aeronautics.

- Westwood, J. (2008). Energy - beyond USD 120 oil. In *All-Energy*, Aberdeen, Scotland.
- Whelan, J. I., Graham, J. M. R., and Peiro, J. (2009). A free-surface and blockage correction for tidal turbines. *Journal of Fluid Mechanics*, 624:281–291.
- Wikipedia (2011). Last accessed on 01/10/2011. http://en.wikipedia.org/wiki/Darrieus_wind_turbine.
- Wilcox, D. (1994). *Turbulence modeling for CFD*. DCW Industries, La Canada, U.S., 2nd edition.