Simulating tidal turbines with multi-scale mesh optimisation techniques

Mohammad Amin Abolghasemi
Submitted for the degree of Doctor of Philosophy and the Diploma of Imperial College

Applied Modelling and Computation Group
Department of Earth Science and Engineering
Imperial College London

September 2016
Abstract

Embedding tidal turbines within simulations of realistic large-scale tidal flows is a highly multi-scale problem that poses significant computational challenges. Herein this problem is tackled using actuator disc momentum (ADM) theory and Reynolds-averaged Navier-Stokes (RANS) with, for the first time, dynamically adaptive mesh optimisation techniques. This enables the mesh to be refined dynamically in time and only in the locations required, thus making optimal use of limited computational resources. Ambient turbulence intensity has a significant effect on the structure of the turbine wake and its recovery. Therefore, both $k - \omega$ and $k - \omega$ SST RANS models have been implemented within the Fluidity framework in order to account for the effects of turbulence. The model is validated against three sets of experiments and a comparison against a similar OpenFOAM model is also presented to portray the benefits of the finite element discretisation scheme employed in the Fluidity ADM-RANS model.

With the aid of the ADM-RANS model, the eddy viscosity value used in depth-averaged models is tuned to improve the wake structures predicted. Thereafter, OpenTidalFarm (OTF) is used to optimise turbine positions in order to maximise the total extracted power from an array. The depth-averaged results are then compared against 3D simulations of the arrays of tidal turbines using the ADM-RANS model, thus allowing for an investigation into the accuracy of the adjoint-based optimisation used in OTF for the first time. Finally, a methodology for embedding the ADM-RANS model within large scale simulations with real tidal forcing and bathymetry is presented. This allows for an accurate prediction of the power output from an array at a realistic tidal site and it can also provide valuable information on how the local and global flows are affected.
Copyright Declaration

The copyright of this thesis rests with the author and is made available under a Creative Commons Attribution Non-Commercial No Derivatives licence. Researchers are free to copy, distribute or transmit the thesis on the condition that they attribute it, that they do not use it for commercial purposes and that they do not alter, transform or build upon it. For any reuse or redistribution, researchers must make clear to others the licence terms of this work.
I hereby certify that the work presented in this dissertation is the result of my own investigations during the PhD project. Text and results obtained from other sources are referenced and properly acknowledged.

Mohammad Amin Abolghasemi
September 2016
They said: “It is not to be found, we too have searched.”
He replied: “That which cannot be found is what I seek.”

Rumi
Acknowledgements

I thank my supervisors for their expert advice, caring support, and inspiration throughout the entire duration of my PhD. Invaluable technical advice, encouragement and overwhelming support from Prof. Matthew Piggott is gratefully acknowledged. My sincere thanks to Dr. Johannes Spinneken for his guidance throughout this work, especially with regards to the experimental work carried out. I would also like to thank Dr. Colin Cotter for his technical advice, especially at the early stages of my PhD. Their brilliance and patience, as well as the independence they allowed me contributed greatly to my personal and professional development.

I would like to thank the entire Applied Modelling and Computation Group (AMCG) at Imperial College London. This vibrant, multidisciplinary and outstanding research group has provided me not only with knowledge and support, but has also helped create memories that I will cherish forever. Namely I would like to thank Dr Stephan Kramer for his invaluable advice and overwhelming support on the numerical work. Moreover, my fellow PhD students and most of AMCG staff who helped me scientifically and more importantly helped me stay positive deserve my gratitude: Alexandros Adam, Dr. Frank Milthaler, Dr. Dave Robinson, Dr. Tim Greaves, Rebecca Smith, Dr. Simon Mouradian and Dr. Alexandros Avdis.

I would also like to thank EPSRC for the industrial CASE award that funded this project. I would like to sincerely thank Sarah Crammond, Ruud Caljouw and David Collier from MeyGen Ltd. for their continual advice and support which helped with the overall direction of this work.
Finally, I would like to thank my parents Masoumeh and Mahmood for their unlimited support and for being an everlasting source of love and happiness in my life. I would also like to thank my family members, Fatemeh, Sobhan, Zahra, Reyhaneh, Mostafa and Asha, all of whom have a warm place in my heart.
# Table of Contents

1 Introduction

1.1 Power of the tide .............................................. 30

1.2 Energy extraction .............................................. 33
   1.2.1 Constrained .............................................. 33
   1.2.2 Unconstrained ........................................... 34

1.3 UK tidal resource .............................................. 35

1.4 Hydrodynamic modelling ...................................... 37
   1.4.1 Blade scale .............................................. 37
   1.4.2 Turbine scale ............................................ 38
   1.4.3 Array scale .............................................. 40
   1.4.4 Regional scale .......................................... 41

1.5 Inner Sound of the Pentland Firth .......................... 42

1.6 Thesis Outline ............................................... 44

2 The ADM-RANS model ........................................... 47

2.1 Introduction .................................................. 47

2.2 Methodology ................................................. 49
   2.2.1 Governing equations ................................. 50
## Table of Contents

2.2.2 Actuator Disc Momentum theory ................................ 51
2.2.3 The $k - \omega$ model ............................................. 52
2.2.4 The $k - \omega$ SST model .......................................... 54
2.2.5 Mesh optimisation .................................................. 56

2.3 ADM verification ...................................................... 57
2.3.1 Semi-analytical solution ......................................... 57
2.3.2 OpenFOAM ADM model .......................................... 58
2.3.3 Results ............................................................ 59

2.4 Conclusions .......................................................... 63

3 ADM-RANS validation: Flow past a porous disc .................. 65
3.1 Introduction .......................................................... 65
3.2 Similitude ............................................................. 67
3.3 Methodology .......................................................... 68
3.3.1 Porous discs ......................................................... 68
3.3.2 Acoustic doppler velocimeter (ADV) .......................... 69
3.3.3 Load cell .......................................................... 70
3.4 Ambient flume conditions .......................................... 72
3.5 Thrust coefficient ..................................................... 73
3.6 Wake profile .......................................................... 75
3.7 ADM-RANS simulation .............................................. 78
3.7.1 Boundary conditions .............................................. 79
3.7.2 ADM-RANS vs experiments ..................................... 81
3.7.3 $C_t$ sensitivity ..................................................... 84
3.7.4 Disc thickness sensitivity ....................................... 85
3.7.5 Mesh optimisation ................................................. 86
3.7.6 Reversing flow ........................................ 90
3.8 Conclusions ................................................. 92

4 ADM-RANS validation: Flow past multiple scaled turbines 93
4.1 Introduction ................................................ 93
4.2 Stallard et al. test case ...................................... 94
  4.2.1 Boundary conditions ..................................... 94
  4.2.2 Ambient conditions ...................................... 95
  4.2.3 Single turbine .......................................... 98
  4.2.4 Three turbines .......................................... 99
  4.2.5 Mesh optimisation ...................................... 102
4.3 Mycek et al. test case ....................................... 107
  4.3.1 Viscosity ............................................... 116
4.4 Conclusions ................................................ 119

5 Investigating the accuracy of tidal turbine array optimisation 121
5.1 Introduction ................................................ 121
5.2 Methodology ............................................... 124
  5.2.1 Depth-averaged model ................................. 124
  5.2.2 3D ADM-RANS model ................................. 126
  5.2.3 Thrust and power calculations ....................... 127
5.3 Viscosity sensitivity ....................................... 129
  5.3.1 Boundary conditions ................................... 130
  5.3.2 Results ................................................. 134
5.4 Channel flow ............................................... 137
  5.4.1 Steady flow .......................................... 138
  5.4.2 Power per turbine ................................... 143
### Table of Contents

5.4.3 Unsteady flow ........................................ 144
5.5 Conclusions ........................................... 150

6 Inner Sound of the Pentland Firth .......................... 153

6.1 Introduction ........................................... 153
6.2 Methodology ........................................... 154
  6.2.1 The domain ....................................... 154
  6.2.2 Absorption term ................................... 157
  6.2.3 Background viscosity ............................... 157
6.3 Results ................................................ 158
  6.3.1 Large-scale flow ................................... 159
  6.3.2 Regional flow ..................................... 162
  6.3.3 Depth profile ...................................... 165
  6.3.4 Power ............................................. 166
6.4 Conclusions ........................................... 166

7 Conclusions ............................................. 169

8 Future work ................................................ 175
  8.1 Experimental ......................................... 176
  8.2 Numerical ............................................. 179

Bibliography ................................................ 181
List of Figures

1.1 The Earth and the Moon orbit about a common barycentre (CM), located within the Earth. The paths of the Earth and the Moon around CM are illustrated via the dotted lines. The centrifugal force (shown in red) due to this orbit has the same magnitude and direction everywhere on the Earth, unlike the Moon’s gravitational force (shown in blue) which depends on the distance to the Moon. The tide producing forces at any point on the Earth are the resultant of these two forces. ......................................................... 31

1.2 The Earth-Sun system. The M2 and the S2 tidal constituents can combine leading to interesting effects. ................................. 32

1.3 Power extraction approaches, taken from Adcock et al. (2015). .... 34

1.4 The mean spring and neap tidal flows around the UK. Source: Department for Business, Enterprise & Regulatory Reform (2008). ... 35

1.5 Potential tidal sites around the UK with tidal stream sites shown on the left and tidal range (both barrage and lagoon) sites shown on the right. The size of the circles reflects the relative sizes of resources in terms of indicative maximum power, not the area which they cover. Source: The Crown Estate (2012)......................................................... 36

1.6 The various scales in hydrodynamic modelling of horizontal axis tidal turbines, adapted from Adcock et al. (2015). ....................... 37

1.7 Water surface elevation of the flow past a 0.4m diameter (1/30th scale) turbine, adapted from Myers and Bahaj (2007). ................. 40
List of Figures

1.8 Map showing MeyGen site location in red. Source: MeyGen Ltd. (2015). .......................................................... 42

1.9 Illustration of a horizontal axis tidal turbine array. Source: MeyGen Ltd. (2015). .................................................. 43

1.10 Schematic of large scale blockage effects in the Inner Sound of the Pentland Firth. Adapted from Tan (2013). ............ 44

2.1 A cut through the tetrahedral Fluidity mesh on the left and the hexahedral OpenFOAM mesh on the right with the disc positioned at the centre. The meshes have been regularly extruded in the streamwise direction with a layer thickness of 1m which is equal to the disc thickness. Note that a finer mesh resolution has been used close to the disc with minimum $l_e/D = 0.10$ in the Fluidity mesh and minimum $l_e/D = 0.02$ in the OpenFOAM mesh. .......................................................... 59

2.2 Numerical domain with $D = 20$ m used in both Fluidity and OpenFOAM simulations corresponding to the comparison plots presented in Fig. 2.3–2.4. The dark grey area represents the actuator disc. .... 59

2.3 Streamwise comparison velocity deficit plots with $Re = 6.0 \times 10^7$. The Conway solution is shown in black ($C_t = 0.2$ (---) and $C_t = 0.8$ (–)) . The Fluidity results are shown in blue ($C_t = 0.2$ (---) and $C_t = 0.8$ (–)) and the OpenFOAM results are shown in red ($C_t = 0.2$ (---) and $C_t = 0.8$ (–)). The dotted lines in (b) correspond to simulation results using a larger computational domain of $25D \times 20D \times 20D$ instead of the usual domain described in Fig. 2.2. .................. 60

2.4 Radial comparison velocity deficit plots with $Re = 6.0 \times 10^7$ at $C_t = 0.2$ and $C_t = 0.8$. The same legend as in Fig. 2.3 is used. ............. 61

3.1 A photograph of the porous disc inside the 4ft flume. .......... 68

3.2 Axial loading on the disc was recorded using a pivoting system. .... 71
### List of Figures

3.3 Boundary layer profile: The water depth is approximately 270 mm but measurements only exist up to 215 mm since the ADV measures the velocity at a point 50 mm below its position and only operates in water. The three profiles shown correspond to one recorded in the centre of the flume and two measurements taken 0.3 m either side of the centre. 72

3.4 Thrust and drag coefficients for the rod and both porous discs. The quadratic fits suggest $C_r = 1.0$ for the rod and that $C_t = 1.08$ and $C_t = 0.91$ for the 45% porous and the 50% porous discs, respectively. The error bars indicate the standard deviation associated with the load cell readings. 74

3.5 Contour plots showing the flow past the 45% porous disc ($C_t = 1.08$) recorded on a plane at hub height behind the disc. $\times$ marks the locations where the ADV was used to record measurements. 76

3.6 ADV time series at two different locations downstream of the porous disc. The raw data is shown in blue and the post-processed data is shown in red. 77

3.7 ADV correlation data at two different locations downstream of the porous disc. The raw data is shown in blue and the post-processed data is shown in red. 77

3.8 Numerical domain with $D = 0.15$ m used in the Fluidity ADM-RANS simulations of the 4ft flume experimental setup with $x$, $y$ and $z$ pointing in the streamwise, lateral and vertical directions, respectively. The dark grey area represents the actuator discs and the light grey area around the discs represents the locations where the additional $\omega$ source term, Eq. (2.9), is applied. 78

3.9 The effect of varying $\omega_{\text{in}}$ on the velocity and turbulence intensity values in the 4ft flume simulations in the absence of any turbines. 80

3.10 Comparison plots along the centreline ($y = 0$, $z = 0$) for the various Fluidity ADM-RANS simulations vs the experimental data. 82

3.11 Lateral comparison plots for the various Fluidity ADM-RANS simulations vs the experimental data at $x = 5D$ and $x = 10D$. The same legend as in Fig. 3.10 has been used. 82
3.12 The TKE downstream of the disc is shown at $2D$, $6D$ and $10D$ downstream of the disc. A ring of TKE forms behind the actuator disc which dissipates away downstream as the wake recovers. 83

3.13 RMSE values in velocity deficit and turbulence intensity for the lateral profiles at $x = 2D, 3D, ..., 12D$ comparing the various ADM-RANS simulations to the experimental data. The same legend as in Fig. 3.10 has been used. 84

3.14 Comparison plots along the centreline ($y = 0, z = 0$) for the Fluidity ADM-RANS simulations vs the experimental data with different $C_t$ values. 84

3.15 Comparison plots along the centreline ($y = 0, z = 0$) for the Fluidity ADM-RANS simulations vs the experimental data with different disc thickness values and $C_t = 1$. 85

3.16 Mesh sensitivity plots along the centreline ($y = 0, z = 0$). 87

3.17 Mesh sensitivity velocity deficit and $I$ plots in the lateral direction at $x = 5D$ and $x = 10D$. The same legend as in Fig. 3.16 has been used. 88

3.18 RMSE values in velocity deficit and turbulence intensity for the lateral profiles at $x = 2D, 3D, ..., 12D$ comparing the different meshes against the mesh converged (i.e. very fine mesh) solution. The same legend as in Fig. 3.16 has been used. 89

3.19 Fluidity ADM-RANS model’s mesh optimisation provides an ideal tool to model reversing flows. 2D slices across the 3D domain are shown at several times throughout the simulation. 91

4.1 Numerical domain for the Stallard et al. experimental setup 95

4.2 Variation of the turbulence intensity, $I$, along the centreline ($y = 0, z = 0$) for different $\omega_m$ values. 96

4.3 Ambient depth profile: Fluidity ADM-RANS vs the experimental data where $x/D = 0$ is the turbine location. 97

4.4 Ambient lateral profile: Fluidity ADM-RANS vs the experimental data where $x/D = 0$ is the turbine location. The same legend as in Fig. 4.3 is used. 97
<table>
<thead>
<tr>
<th>Figure</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>4.5</td>
<td>Flow past a single turbine. Lateral velocity deficit profiles showing the ADM-RANS $k - \omega$ with correction terms results at 8D (-----) and 12D (....... and the $k - \omega$ SST results at 8D (- -) and 12D (....) against the self-similar solution (---). The experimental results at 8D (--) and 12D (--) are also shown.</td>
</tr>
<tr>
<td>4.6</td>
<td>Stallard et al. test case: Comparison plots along the centreline</td>
</tr>
<tr>
<td>4.7</td>
<td>Lateral velocity deficit, $1 - u_x/u_0$, profiles for the ADM-RANS simulations vs Stallard et al. at $x = 2D, 4D, 6D, 8D, 10D$ and $12D$ with $z = 0$. The same legend as in Fig. 4.6 is used.</td>
</tr>
<tr>
<td>4.8</td>
<td>Lateral turbulence intensity, $I$, profiles for the ADM-RANS simulations vs Stallard et al. at $x = 2D, 4D, 6D, 8D, 10D$ and $12D$ with $z = 0$. The same legend as in Fig. 4.6 is used.</td>
</tr>
<tr>
<td>4.9</td>
<td>RMSE values in velocity deficit and turbulence intensity for Fig. 4.7–4.8. The same legend as in Fig. 4.6 is used.</td>
</tr>
<tr>
<td>4.10</td>
<td>A cut through the mesh used for the fixed mesh simulation with the discs positioned in the centre. A finer resolution of $l_e/D = 0.055$ is used around the discs with a coarse resolution of $l_e/D = 0.370$ used in the periphery. This mesh has been regularly extruded in the streamwise direction with a layer thickness of $D/8$ (0.03375 m) which is equal to the disc thickness.</td>
</tr>
<tr>
<td>4.11</td>
<td>Lateral velocity deficit and turbulence intensity plots at $x = 2D$, $x = 6D$ and $x = 12D$ with $z = 0$ showing the results from both the fixed mesh and the adaptive mesh simulations using the ADM-RANS $k - \omega$ SST model with $C_t = 0.92$.</td>
</tr>
<tr>
<td>4.12</td>
<td>Results from an ADM-RANS $k - \omega$ SST simulation of the Stallard et al. experimental setup is shown. During the ADM-RANS simulation dynamic mesh optimisation has been used to adapt to the velocity, $k$ and $\omega$ fields. A 2D slice across the 3D domain is presented here showing the velocity and $k$ fields at hub height ($z = 0$) along with the adapted mesh.</td>
</tr>
<tr>
<td>4.13</td>
<td>Numerical domain for the Mycek et al. experimental setup</td>
</tr>
</tbody>
</table>
4.14 Results from an ADM-RANS $k - \omega$ SST simulation of the Mycek et al. experimental setup is shown. A 2D slice across the 3D domain is presented here showing the velocity and $k$ fields at hub height ($z = 0$) along with the adapted mesh for the $I_\infty = 3\%$ case with $C_t = 0.81$ and $u_{in} = 0.80 \text{ m s}^{-1}$. .................................................. 109

4.15 Results from an ADM-RANS $k - \omega$ SST simulation of the Mycek et al. experimental setup is shown. A 2D slice across the 3D domain is presented here showing the velocity and $k$ fields at hub height ($z = 0$) along with the adapted mesh for the $I_\infty = 15\%$ case with $C_t = 0.75$ and $u_{in} = 0.83 \text{ m s}^{-1}$. .................................................. 110

4.16 Lateral velocity deficit and turbulence intensity profiles for the ADM-RANS simulations vs Mycek et al. at $x = 2D, 4D, 6D, 8D$ and $10D$ for the upstream turbine with $I_\infty = 3\%$, $C_t = 0.81$ and $u_{in} = 0.80 \text{ m s}^{-1}$. The same legend as in Fig. 4.6 is used. ....................... 111

4.17 Lateral velocity deficit and turbulence intensity profiles for the ADM-RANS simulations vs Mycek et al. at $x = 2D, 4D, 6D, 8D$ and $10D$ for the upstream turbine with $I_\infty = 15\%$, $C_t = 0.75$ and $u_{in} = 0.83 \text{ m s}^{-1}$. The same legend as in Fig. 4.6 is used. ....................... 112

4.18 Lateral velocity deficit and turbulence intensity profiles for the ADM-RANS simulations vs Mycek et al. at $x = 2D, 4D, 6D, 8D$ and $10D$ for the downstream turbine with $I_\infty = 3\%$, $C_t = 0.81$ and $u_{in} = 0.80 \text{ m s}^{-1}$. The same legend as in Fig. 4.6 is used. ....................... 113

4.19 Lateral velocity deficit and turbulence intensity profiles for the ADM-RANS simulations vs Mycek et al. at $x = 2D, 4D, 6D, 8D$ and $10D$ for the downstream turbine with $I_\infty = 15\%$, $C_t = 0.75$ and $u_{in} = 0.83 \text{ m s}^{-1}$. The same legend as in Fig. 4.6 is used. ....................... 114

4.20 RMSE values in velocity deficit and turbulence intensity for Fig. 4.16–4.19. The same legend as in Fig. 4.6 is used. ....................... 115

4.21 A 2D slice through the 3D domain showing the variation of viscosity ($\nu_{bg} + \nu_T$) for the $k - \omega$ (top), $k - \omega$ with correction terms (middle) and the $k - \omega$ SST (bottom) simulations. ....................... 116
4.22 A 2D slice through the 3D domain showing the variation of turbulence length scale for the $k - \omega$ (top), $k - \omega$ with correction terms (middle) and the $k - \omega$ SST (bottom) simulations.

5.1 Numerical domain with turbine diameter $D = 18.75$ m and thickness $D/4$ used in the 3D ADM-RANS simulations. The dark grey area represents the actuator disc positioned at mid-depth. In the OTF simulations an equivalent 2D depth-averaged domain was used with a $D \times D$ square turbine drag area.

5.2 Variation of $\nu_T$ at mid-depth for the 3D ADM-RANS model with the domain used shown in Fig. 5.1.

5.3 Lateral velocity deficit profiles at $6D$ and $12D$ at four different $u_{in}$ values for the OTF and the Fluidity ADM-RANS simulations. The dotted lines represent the OTF results where $\nu$ is increased from $0.1 \text{ m}^2\text{s}^{-1}$ to $1 \text{ m}^2\text{s}^{-1}$. The 3D ADM-RANS results have been depth-averaged and are plotted on top of the OTF results to help identify the OTF $\nu$ value that best matches the ADM-RANS profiles.

5.4 Based on the wake profiles presented in Fig. 5.3, the OTF $\nu$ value that provides the closest match to the ADM-RANS profile has been selected for each ADM-RANS simulation and the results are presented here. The dotted lines show the linear relationship described in Eq. (5.19).

5.5 OTF domain for an array of 32 turbines in an ideal channel where the grey area represents the site where turbines can be positioned.

5.6 OTF results showing the initial staggered layout (12.98 MW) on the left and the optimised layout (17.12 MW) on the right for the $u_{in} = 1.9 \text{ m s}^{-1}$ case.

5.7 OTF results showing the initial staggered layout (45.48 MW) on the left and the optimised layout (61.93 MW) on the right for the $u_{in} = 2.9 \text{ m s}^{-1}$ case.
5.8 Iso-surfaces of constant speed, $||\mathbf{u}|| = 0.85 u_{in}$, are shown here to illustrate the wakes formed behind the turbine array in both the staggered and optimised layouts for the Fluidity ADM-RANS simulations with $u_{in} = 2.9 \text{ m s}^{-1}$. A 2D slice at hub height through the 3D domain is also shown to demonstrate the optimised mesh where a higher resolution is used near the turbines and their wakes. The turbine colours correspond to the relative power output with red being the highest and blue corresponding to the lowest.

5.9 Total array power output at each OTF optimisation iteration. Iteration 1 represents the power output from the initial staggered layout. The crosses correspond to the power estimate obtained from a 3D Fluidity ADM-RANS simulation on the corresponding array layout.

5.10 OTF vs. Fluidity ADM-RANS power per turbine comparison plot for the staggered and optimised layouts. The diagonal line indicates a perfect match between the two models.

5.11 OTF results showing the initial staggered layout on the left and the optimised layout on the right for the time-dependent inflow velocity case.

5.12 Variation of the inlet boundary conditions with time for the unsteady flow ADM-RANS simulations during the 6 h half-cycle.

5.13 2D slices at hub height through the 3D mesh at various times of the ADM-RANS unsteady flow simulation past the optimised layout.

5.14 OTF vs ADM-RANS unsteady flow. The solid lines show the ADM-RANS results (left (____) and right (____)) and the dashed lines show the OTF results with a variable viscosity (left (___) and right (___)). The dotted lines show OTF results with $\nu = 3 \text{ m}^2\text{s}^{-1}$ (left (_____)) and right (_____). The shaded areas represent the range of values obtained if constant eddy viscosity values of $0.1 \text{ m}^2\text{s}^{-1} \leq \nu \leq 1 \text{ m}^2\text{s}^{-1}$ are used in OTF.
<table>
<thead>
<tr>
<th>Fig.</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>6.1</td>
<td>Domain considered in the Inner Sound simulations. Taken from Martin-Short et al. (2015). Map of the UK is shown in (A) and the simulation domain is shown in (B). A zoom-in of the Pentland Firth and the surrounding islands is shown in (C). The colour relief represents the resampled bathymetry used here.</td>
</tr>
<tr>
<td>6.2</td>
<td>The unstructured surface mesh used in the Inner Sound simulations. A higher resolution is used near coastlines and at the array site. The entire domain surface mesh is shown in (a) with a zoom-in of the Pentland Firth shown in (b). A zoom-in of the Inner Sound mesh is shown in (c) demonstrating how the resolution increases at the array site and a zoom-in of the structured array site surface mesh is shown in (d). The locations of the four turbines is also shown in red.</td>
</tr>
<tr>
<td>6.3</td>
<td>A 2D slice across the 3D mesh showing how the number of layers increases at the array site. The turbine (actuator disc) is shown in red.</td>
</tr>
<tr>
<td>6.4</td>
<td>A variable $\nu_{bg}$ was used in the Inner Sound simulations that gradually increases from a value of $1 \times 10^{-6} \text{m}^2\text{s}^{-1}$ within the array site to a value of $1 \text{m}^2\text{s}^{-1}$ everywhere else.</td>
</tr>
<tr>
<td>6.5</td>
<td>Inner Sound velocity</td>
</tr>
<tr>
<td>6.6</td>
<td>Inner Sound $\nu_T$</td>
</tr>
<tr>
<td>6.7</td>
<td>2D slices of the 3D domain at turbine hub height for the $k-\omega$ SST simulation showing $I$ variation within the Inner Sound.</td>
</tr>
<tr>
<td>6.8</td>
<td>Array site velocity</td>
</tr>
<tr>
<td>6.9</td>
<td>Array site $\nu_T$</td>
</tr>
<tr>
<td>6.10</td>
<td>2D slices of the 3D domain at turbine hub height for the $k-\omega$ SST simulation showing the $I$ variations within the array site.</td>
</tr>
<tr>
<td>6.11</td>
<td>2D slices of the 3D domain at turbine hub height for the $k-\omega$ SST simulation showing the $k$ variations within the array site.</td>
</tr>
<tr>
<td>6.12</td>
<td>Depth profiles showing the flow past turbine 2 at $t = 6.67 \text{h}$ for the $k-\omega$ SST simulation.</td>
</tr>
<tr>
<td>6.13</td>
<td>Power outputs per turbine during the Inner Sound simulations.</td>
</tr>
<tr>
<td>6.14</td>
<td>Total power output during the Inner Sound simulations.</td>
</tr>
</tbody>
</table>
List of Figures

8.1 An illustration of a 32 turbine array layout inside the new wide flume currently under construction. .................................................. 177

8.2 The depth profile inside the flume can be altered by covering the flume bed with marbles. .................................................. 178

8.3 Boundary layer profile: Normal flume vs flume covered with marbles measured at the centre of the flume. .......................... 178
List of Tables

3.1 Experimental scale of the 4ft flume experiments where a 1:125 scale is used. ................................. 67

3.2 Porous discs properties. .............................................. 69

3.3 Properties of the different meshes used. *These are the maximum values use in the adaptive simulation. ......................... 86

3.4 RMSE values in velocity deficit and turbulence intensity comparing the various simulations to the mesh converged (i.e. very fine mesh) solution. The corresponding wake profiles are shown in Fig. 3.16. . . . 88

4.1 RMSE values in velocity deficit and turbulence intensity comparing the fixed and adaptive mesh simulation results to the experimental data. The corresponding wake profiles are shown in Fig. 4.11. . . . . 103

4.2 Fluidity ADM-RANS simulation inlet conditions for the Mycek et al. test cases. The velocity values are taken from Mycek et al. (2014a) and the $k$ values are chosen to give turbulence intensity values similar to those recorded experimentally. .......................... 108

4.3 Maximum RMSE values in velocity deficit and turbulence intensity comparing the simulation results to the experimental data. The corresponding wake profiles are shown in Fig. 4.16–4.19. The RMSE plots are shown in Fig. 4.20. ...................... 115
List of Tables

5.1 Different inlet conditions for scenario (a) of the ADM-RANS simulations. .......................... 133
5.2 Different inlet conditions for scenario (b) of the ADM-RANS simulations. .......................... 133
5.3 Different inlet conditions for scenario (c) of the ADM-RANS simulations. .......................... 133
5.4 Steady flow simulation array power values comparing the variable viscosity OTF model against the 3D ADM-RANS model. .......................... 142
5.5 Unsteady flow simulation power integral values. The percentage difference of the OTF data relative to the ADM-RANS data is shown inside the brackets. .......................... 148
Chapter
ONE

Introduction

Global warming concerns and anxiety over fossil fuel reserves and supply security have resulted in a significant shift towards renewable energy sources in the past decade. The UK has set targets to produce 15% of its energy from renewable sources by 2020 (Department of Energy & Climate Change, 2011). This is a substantial increase considering that only 4.1% of the country’s energy needs came from renewable sources in 2012 (Department of Energy & Climate Change, 2013). The oceans surrounding the UK offer a vast resource of largely untapped energy which can be extracted. Hence, marine renewable energy (MRE) has the potential to play a vital role in helping achieve this target. However, in order to maximise its potential and reduce the cost of energy, detailed investigations into the optimisation of MRE extraction techniques are essential.

There has already been a great increase in the number of offshore wind turbine farms around the UK. Between July 2012 and June 2013 offshore wind capacity increased by 1 GW, bringing the total installed capacity to 3.5 GW. Furthermore, generation rose to 9.7 TW h for the year July 2012 to June 2013, increasing by 3.5 TW h on the year before (Department of Energy & Climate Change, 2013). Wave and tidal energy are also on the ascendancy and have attracted a significant amount of interest over the past decade. This sector is still in the developmental stage and
is yet to make a substantial contribution to the UK’s energy production. However, there are a number of exciting projects currently underway, especially in the tidal sector, which will undoubtedly help reduce the UK’s dependency on fossil fuels.

This study focuses on tidal power, specifically tidal stream, and its potential to become a reliable renewable energy source. In this introductory chapter, initially a brief description of the mechanisms influencing the global tide is presented. Thereafter, the different approaches used to extract energy from the tide are described. This is followed by focusing on the UK and outlining the great potential for tidal power which the UK benefits from. Furthermore, the hydrodynamic modelling challenges faced are outlined and the MeyGen tidal stream project in the Inner Sound of the Pentland Firth is introduced. Finally, the chapter concludes with an outline of the rest of the thesis.

1.1 Power of the tide

Tides are a result of the orbital mechanics of the Earth, the Moon and the Sun. One of the earliest explanations for this was provided in the Principia by Newton in what is known as the *Equilibrium Theory of Tides*. In this simplified model, the Earth and the Moon orbit their common centre of mass (CM), also known as the barycentre, with the Earth assumed to be a perfect sphere covered by oceans, Fig. 1.1. Due to the major difference in the mass of the two objects in the Earth-Moon system, where $m_{\text{moon}} = 0.0123 m_{\text{earth}}$, the barycentre is located within the Earth (The Open University, 1999).

This orbit results in a centrifugal force on the Earth parallel to the line joining the centres of the Earth and the Moon which is equal in magnitude and direction everywhere on the Earth as every point follows the dotted path shown in Fig. 1.1. On the other hand, the gravitational attraction of the Moon is also acting on the Earth, but unlike the centrifugal force, the magnitude and direction of this force varies with distance from the Moon. The resultant of these two forces is known as the *tide producing force*. For example, at point A, since the gravitational attraction of the Moon is greater than the centrifugal force, the tide producing force will point towards the Moon, and a tidal bulge appears. Vice versa, at point B, the gravita-
1.1: Power of the tide

Figure 1.1: The Earth and the Moon orbit about a common barycentre (CM), located within the Earth. The paths of the Earth and the Moon around CM are illustrated via the dotted lines. The centrifugal force (shown in red) due to this orbit has the same magnitude and direction everywhere on the Earth, unlike the Moon’s gravitational force (shown in blue) which depends on the distance to the Moon. The tide producing forces at any point on the Earth are the resultant of these two forces.

The tidal attraction of the Moon is weaker than the centrifugal force, hence the tide producing force will point away from the Moon, which leads to another tidal bulge.

As the Moon orbits the Earth, these bulges will follow the Moon, and since the Earth is also rotating about its own axis, this leads to the principal lunar semi-diurnal (M2) tidal constituent with a tidal cycle of 12 hours and 25 minutes. The reason for this lies in the fact that the Moon orbits the Earth in the same direction as the Earth rotates around itself. Therefore, it takes the Moon slightly longer than a day to be above the same location on the Earth.

Similarly, the Sun’s gravitational pull also leads to tidal constituents on the Earth, but these are typically not as strong as those due to the Moon given that it is significantly further away. These are known as the principal solar semi-diurnal (S2) tidal constituent and have a tidal cycle of 12 hours. Although, the S2 tidal constituent is not as strong as the M2 tidal constituent, the two semi-diurnal constituents can combine leading to interesting effects. When the Sun and the Moon are in line with the Earth, i.e. in phase, the superposition of the solar and lunar tides leads to the stronger spring tides with a cycle of around two weeks. When the Moon is out of phase by $\pi/2$ or $3\pi/2$, the solar influence weakens the lunar tide, leading to the weaker neap tides, Fig. 1.2.
Chapter 1: Introduction

(a) When the Moon and the Sun are on the same side of the Earth, or exactly opposite, the solar tides (dark blue) and the lunar tides (light blue) are in phase resulting in the higher spring tides.

(b) When the Moon and the Sun are out of phase by $\pi/2$, or $3\pi/2$, the solar tides (dark blue) act against the lunar tides (light blue) resulting in the lower neap tides.

Figure 1.2: The Earth-Sun system. The M2 and the S2 tidal constituents can combine leading to interesting effects.

The actual tide is a result of the superposition of many tidal constituents, each corresponding to a particular astronomical motion of the Earth, the Moon or the Sun, only two of which have been discussed here.

In reality, the observed tides are not in perfect agreement with the predicted equilibrium tides. There are a number of reasons for this:

- the variations in the Earth’s bathymetry leads to variations in the speed of the tidal waves travelling around the globe.

- the presence of land masses inevitably alters the path of tidal flows.
1.2: Energy extraction

- the Earth’s Coriolis force acts orthogonally to the right of the flow in the Northern Hemisphere and orthogonally to the left of the flow in the Southern Hemisphere and this will enhance the tide driving forces in some regions and diminish them in other parts.

- a time-lag develops since the Earth’s rate of rotation is too rapid for the inertia of the water masses to establish an equilibrium tide immediately.

Hence, the exact nature of tidal flows is more complex than the simplified equilibrium theory explained above. However, given that astronomical motions forming the system can be predicted well in advance and the Earth’s bathymetry is reasonably documented, there is a thorough understanding of global tidal flows. Therefore, tidal motions around the globe are highly predictable and this is a huge advantage for energy production, especially compared to other sporadic and unpredictable MRE resources such as wind power.

1.2 Energy extraction

The most efficient method to extract energy from the tide is to force the water to pass through a turbine. In this section, the various approaches are categorised into two different groups, similar to the study of Adcock et al. (2015), and each group is briefly discussed in turn.

1.2.1 Constrained

In this approach the water is constrained by a barrier and all the water has to pass through this barrier during power generation. Hence, a significant head difference can be built up across the barrier, Fig. 1.3a. If the barrier is built across an estuary, it is known as a tidal barrage and if it exists completely off-shore, or simply closes off a tidal sea area, it is named a tidal lagoon.

The prime example of a tidal barrage is the tidal power station at La Rance in northern France which was opened in 1966. This power station houses 24 10 MW turbines with an average annual output of around 500 GW h (Andre, 1978) and has
Chapter 1: Introduction

Figure 1.3: Power extraction approaches, taken from Adcock et al. (2015).

been operating successfully for half a century. The greatest concern with tidal barrages is their negative impact on the environment. One of the biggest issues at La Rance was that during construction the estuary was isolated from the ocean, resulting in major environmental damage that virtually eradicated the marine fauna and flora (Retiere, 1994). Hence, this methodology is unlikely to be followed in future projects and the dam would most likely be constructed off-site and installed in stages. This will limit impacting the environment during construction, but there will still be changes to the environment during operation of the barrage. These include variations in salinity and water quality, alterations to sediment transport and disruption in the movement of fish and marine mammals (Retiere, 1994; Adcock et al., 2015).

Alternatively, the water can be only locally constrained and not all the flow has to pass through the barrier, Fig. 1.3b. A head difference will still form across the barrier, but the negative impact on the environment during operation is reduced. This approach is more suitable near headlands where the flow is locally accelerating. However, this will need to be happening on a large enough scale to make the project economically viable given the huge costs associated with constructing tidal barriers in oceans.

1.2.2 Unconstrained

In this design, the flow is not forced to pass through a barrier and instead turbines are positioned in locations with a high tidal flow velocity. This approach is known as tidal stream power. Here, the flow is free to pass either side, or even above or below, the turbines, Fig. 1.3c and the turbines only generate power when the local
velocity is above the \textit{cut-in} velocity of the turbine. In this approach, the flow is still constrained to pass through the turbines, however the level of constrain applied on the flow is significantly lower than the methods described in section 1.2.1. Furthermore, the environmental effects associated with this approach are expected to be much lower since there will be minimum restriction to the movement of fish and marine mammals. However, there will still be changes to the local flow and this will affect sediment transport in the region (Martin-Short et al., 2015).

\section*{1.3 UK tidal resource}

\begin{figure}[h]
\centering
\begin{subfigure}[h]{0.45\textwidth}
\includegraphics[width=\textwidth]{mean_spring_tide}
\caption{Mean spring tide.}
\end{subfigure} \hfil
\begin{subfigure}[h]{0.45\textwidth}
\includegraphics[width=\textwidth]{mean_neap_tide}
\caption{Mean neap tide.}
\end{subfigure}
\caption{The mean spring and neap tidal flows around the UK. Source: Department for Business, Enterprise & Regulatory Reform (2008).}
\end{figure}

The UK benefits from a vast resource of largely untapped tidal energy which can be available to help achieve the 2020 European Union Renewable Energy Directive and future targets. In 2008, the Department for Business, Enterprise and Regulatory Reform (BERR) published the Atlas of UK Marine Renewable Energy Resources where real bathymetry data was used to analyse the tidal flows around the UK. Therein, mean tidal flows around the UK for both the stronger \textit{spring tides} and
the weaker *neap tides* were estimated, Fig. 1.4. This indicated that the majority of the resources are located in Scottish waters, around Wales and the Southwest of England. Consequently, potential power generation sites could be identified.

![Figure 1.5: Potential tidal sites around the UK with tidal stream sites shown on the left and tidal range (both barrage and lagoon) sites shown on the right. The size of the circles reflects the relative sizes of resources in terms of indicative maximum power, not the area which they cover. Source: The Crown Estate (2012).](image)

In a recent study published in 2012 by The Crown Estate the potential power generation from tidal resources was examined in greater detail. The theoretical UK tidal resource was estimated to be 216 TWh/year. This estimate assumes 96 TWh/year from tidal barrages, 25 TWh/year from tidal lagoons and 95 TWh/year from tidal stream turbines (The Crown Estate, 2012). A detailed outline of the locations of the potential power generation sites was produced and this is presented in Fig. 1.5. The majority of the tidal stream sites are concentrated around the Orkney islands and in the Irish Sea. The largest single area of tidal range resource is located in the Bristol Channel and the Severn Estuary. Note that the power values quoted are theoretical estimates based on a preliminary study and only outline the huge potential available. The ultimate power generation values are likely to be a lot lower given that these
values are solely based on the kinetic energy flux of the flow and that existing sea
uses and environmental factors were not considered in the aforementioned study.

1.4 Hydrodynamic modelling

In this study, the focus is on tidal stream power. Embedding tidal turbines within
simulations of realistic large-scale tidal flows is a highly multi-scale problem that
poses significant computational challenges. The various scales are introduced be-
low following a similar classification to that presented by Adcock et al. (2015). A
schematic of the different length scales is presented in Fig. 1.6.

Figure 1.6: The various scales in hydrodynamic modelling of horizontal axis tidal turbines,
adapted from Adcock et al. (2015).

1.4.1 Blade scale

This is the finest scale considered and focuses on the flow around the turbine blades
and the corresponding lift and drag. In order to do so the boundary layer has to be
resolved leading to huge computational costs for the numerical simulations. This
field is more of an interest to turbine manufacturers and involves similar practices
carried out in the wind and aeronautics industry. It is not of a primary importance
in this study.
1.4.2 Turbine scale

As the fluid passes through the tidal turbine, it applies a torque onto the rotor blades enabling the kinetic energy of the flow to be converted into electrical power. This results in the development of a low pressure region behind the turbine; i.e. the wake. In the near wake region, a few diameters downstream of the turbine, the flow exhibits high turbulence levels mainly generated by the axial velocity shear, blade related vortices and wake rotation (Roc et al., 2013). This area of reduced axial momentum would typically last several diameters downstream until the undisturbed flow mixes in with the fluid in the wake, re-energising the wake. Hence, the flow gradually returns to its uniform state making it suitable for the installation of another turbine downstream.

At the turbine scale, the factors affecting the wake profile are explored. Some of these factors, such as thrust coefficient, which itself depends on blockage, affect wake generation. Other factors such as ambient turbulence intensity and proximity to the free surface and the sea bed affect the rate of wake recovery.

1.4.2.1 Turbulence intensity

One of the main challenges when attempting to simulate the flow past a tidal turbine is the ability to correctly account for the turbulence within the flow. Previous studies have indicated that the ambient turbulence intensity \( I \) has a significant effect on the structure of the turbine wake and its recovery. Blackmore et al. (2014) demonstrated how varying the turbulence intensity in a gravity based flume affects the drag forces acting on solid and porous disc rotor simulators. The drag coefficients varied by as much as 20% as the turbulence intensity was increased from 4.5% to 13%.

Moreover, Mycek et al. (2014a) explored the wake profile downstream of a scaled turbine for two different turbulence intensities of 3% and 15%. It was demonstrated that for the \( I = 3\% \) case, the velocity along the centreline 8 diameters downstream of the turbine had recovered to around 60% of the unperturbed value, whereas for the \( I = 15\% \) case the velocity was close to 90% of this value. Hence, it is crucial that the hydrodynamic model is able to account for these differences and correctly
quantify their effect on the rate of wake recovery.

1.4.2.2 Local blockage

Myers and Bahaj (2007) examined the flow past a 0.4 m diameter (1/30th scale) turbine in a circulating water channel. An increase in free surface height was observed immediately upstream of the turbine due to local blockage as the water slowed down on approach to the turbine. Local blockage, $B_L$ is typically defined (Nishino and Willden, 2012b) as

$$B_L = \frac{\text{single device area}}{\text{local passage cross-sectional area}}. \quad (1.1)$$

Immediately downstream, this obstruction to the flow is no longer present and the flow accelerates resulting in a decrease in free surface height. In the far wake (typically 6-8 diameters downstream) as the wake expands, the free surface height increases again and returns to the unperturbed values observed upstream and away from the turbine, Fig 1.7. The free surface height will always increase as the wake expands, however it does not always return to the unperturbed values (Adcock et al., 2015).

Myers and Bahaj (2007) suggest that in a realistic environment, such extreme variations are unlikely since the Froude numbers will be lower. The Froude number ($Fr$) is defined as the ratio of the characteristic fluid velocity to the shallow water wave velocity,

$$Fr = \frac{u}{\sqrt{gd}} \quad (1.2)$$

where $u$ is the characteristic flow velocity, $d$ is water depth and $g$ is the acceleration due to gravity. Hence, in deeper flows the changes in free surface height as a result of the extraction in energy are expected to be lower than the values observed at the model scale. The free surface height is not always included in numerical studies, but in high $Fr$ cases their effect is not negligible and the hydrodynamic model needs to account for this.

Although free surface variations can be overlooked in certain cases, local flow ac-
1.4.3 Array scale

This scale deals with the interaction of the wakes within an array of turbines and how this changes by varying the inter-turbine spacing. At the turbine scale, the main aim is to develop a model which can correctly account for all the major factors that affect the wake. However, if the model is to be used to assess the flow past arrays of turbines, it needs to be of a lower-fidelity to make larger scale analysis feasible. Hence, at the array scale, depending on the aims of the study, only the most important factors are focused on. Given the close link between this scale and the turbine scale, there is a lot of overlap between the two, as indicated in Fig. 1.6. Hence, many models developed for the turbine scale have been extended and used to model flow past arrays of tidal turbines. Furthermore, in order to account for the effect of flow blockage at this scale, the concept of global blockage, $B_G$ is introduced (Nishino and Willden, 2012b) where

$$B_G = \frac{\text{total device area}}{\text{channel cross-sectional area}}. \quad (1.3)$$

At the array scale, rather than focusing on the wake behind a single turbine, the array wakes and the effect of one row of turbines on another row is investigated. Hence, the effect of array layout on power production and environmental impacts can be studied. For example, Divett et al. (2013) developed a 2D depth-averaged
model to investigate the flow past an array of 15 turbines inside an ideal channel arranged in 5 rows of three. Therein, it was demonstrated that by changing the array layout from in-line to staggered, a 54% increase in the array power production can be achieved. The reason for this lies in the fact that by changing to a staggered layout the upstream velocity for each turbine is increased and therefore the rate of momentum extraction can be increased leading to greater power production values.

In a recent study, Masters et al. (2015) investigated the performance of a range of computational models of horizontal axis tidal turbines at different spatial scales. As part of that study, both a high resolution 3D blade element momentum (BEM) model and a large scale 2D depth-averaged model were used to simulate the flow past a small tidal turbine array at an idealised headland. It was demonstrated that while the flow velocities in the far upstream were very similar, substantial differences exist in the wake profiles of the two models. This is not surprising given the differences in spatial and temporal scales used, as well as the different treatment of turbulence in the two approaches. Ultimately, it was concluded that the choice of model will depend on the physical scale of interest and the computational resources available.

### 1.4.4 Regional scale

At the regional scale, the real bathymetry and local coastline is taken into account to examine how the flow at a specific site changes due to the presence of an array of turbines. Furthermore, this scale implements real tidal forcing driving the tidal currents through the turbines in order to look into how the tidal flows around the region where the array is located will be influenced. This scale will typically take into account a much larger domain than that considered during an array scale study. In order to numerically simulate arrays of turbines, these models usually adopt an approach where the turbines are represented as a region of increased bottom drag (Sutherland et al., 2007; Divett et al., 2013; Funke et al., 2014; Martin-Short et al., 2015).

For example, Martin-Short et al. (2015) investigated the potential impacts of large arrays of tidal turbines on the flow regime and sediment transport in the Inner Sound of the Pentland Firth. Therein, a multi-scale 2D depth-averaged model where the resolution varied from 18 m to 20 km was used and the turbines were represented
as regions with enhanced bottom drag. The tidal forcing applied to drive the tidal currents was taken from recorded values and real bathymetry data was used in order to present an accurate picture of the local tidal flow variations within the Inner Sound. Furthermore, the model was validated against tidal gauges and ADCP data from the Inner Sound and was shown to be in good agreement with sea surface elevation changes and current velocities. Martin-Short et al. (2015) demonstrated the significant tidal asymmetry experienced by the Inner Sound which results in the net movement of sediment from west to east under natural conditions. Arrays of tidal turbines in excess of 85 MW were shown to affect this natural pattern of sediment migration. The numerical model was used to demonstrate that as the flow is diverted around the arrays and the most favourable region for sediment accumulation moves to the centre of the array. This may lead to long term accumulations of gravel and coarse sand at the centre, as well as scouring and removal of existing sediment deposits to the north and south of the array.

1.5 Inner Sound of the Pentland Firth

This PhD is a collaboration with MeyGen Ltd focused on the Inner Sound region of the Pentland Firth, a narrow channel of water that separates the north Scottish mainland from Stroma Island in the Orkneys, Fig. 1.8. This site has an excellent tidal resource and has been chosen due to its high tidal flow rate and relatively shallow depth. The MeyGen project will have a maximum aggregated capacity in
excess of 398 MW after the final phase has been completed. Although the exact plan is yet to be finalised, it has been decided that the 398 MW rated capacity will be achieved using horizontal axis turbines similar to the ones shown in the illustration in Fig. 1.9. The turbines will be able to rotate about the vertical axis (i.e. yaw) in order to face incoming flow as the tide changes from flood to ebb.

![Figure 1.9: Illustration of a horizontal axis tidal turbine array. Source: MeyGen Ltd. (2015).](image)

As more and more turbines are installed in a channel, the risk of choking the flow and not extracting the full power available increases. Hence, the effects of flow blockage need to be carefully considered. In the Inner Sound as the number of turbines installed increases the average flow velocity through the site reduces and more flow will be diverted north of Stroma, Fig. 1.10a. Similarly, within the Inner Sound the flow accelerates in regions of low resistance and decelerates in regions of high resistance, where turbines are present, and the flow will be encouraged to go around the array of tidal turbines, Fig. 1.10b.

In order to maximise the power output of arrays of tidal turbines, the velocity profiles and changes in turbulence levels across an array of turbines, as well as the local, regional and global blockage effects, must be studied in depth. The ultimate aim is to obtain a better understanding of these array effects in order to help produce an optimal turbine layout for a realistic tidal site, such as the Inner Sound.
Chapter 1: Introduction

As the number of turbines installed increases the average flow velocity through the site reduces and more flow will be diverted north of Stroma.

The flow accelerates in regions of low resistance and decelerates in regions of high resistance and the flow will be encouraged to go around the array of tidal turbines.

Figure 1.10: Schematic of large scale blockage effects in the Inner Sound of the Pentland Firth. Adapted from Tan (2013).

1.6 Thesis Outline

This study focuses on the extraction of tidal stream energy from coastal waters via horizontal axis tidal turbines. A brief introduction to the global tide and the various power generation approaches available was presented and the UK’s tidal resources have been briefly discussed. The multi-scale nature of the problem was outlined along with the various approaches used for modelling and investigating the flow past horizontal axis tidal turbines. A summary of the MeyGen project in the Inner Sound of the Pentland Firth was also provided.

The original aims and objectives of the thesis are:

- developed a numerical tool that can bridge the gap between 3D turbine scale models and 2D depth-averaged regional scale models
- account for the effects of ambient turbulence intensity on the wake structures
- validate the model against a series of experimental flume test results
- use the model to investigate the limitations associated with using 2D depth-averaged regional scale models
- use the model to investigate the effects of placing arrays of tidal turbines in the Inner Sound of the Pentland Firth
The rest of the thesis is organised as follows. Firstly, in chapter 2, the 3D ADM-RANS model developed in Fluidity is presented alongside a verification against the Conway (1995) analytical model and a similar OpenFOAM model. Furthermore, a description of the turbulence models developed in Fluidity and the mesh optimisation techniques used is provided. Thereafter, in chapter 3 an experimental analysis of the flow past a porous disc is presented. This will outline the shortcomings of the numerical model as a result of the assumptions used in the development of the Fluidity ADM-RANS model. Further validation against multiple scaled turbines is provided in chapter 4 where the performance of the ADM-RANS model, at the turbine scale, is assessed against two sets of experimental flume test results. The suitability of the mesh optimisation techniques used is also assessed in detail. Chapter 5 focuses on the array scale and the ADM-RANS model is used to assess the sensitivity to the viscosity coefficient used in depth-averaged models. Consequently, the accuracy of the adjoint-based optimisation used in the OpenTidalFarm package, which uses a depth-averaged model, is investigated. Finally in chapter 6, the ADM-RANS model is embedded in a regional scale simulation of the Inner Sound to assess the effect of a small array of 4 tidal turbines in a realistic scenario. The thesis concludes with a Conclusions section reflecting on the main findings of the study. The Future work section outlines how this study can be taken forward.
Chapter

TWO

The ADM-RANS model

This chapter is derived from and expands upon a journal paper published in the Journal of Fluids and Structures by Abolghasemi et al. (2016b). The material therein has arisen as the result of the first author’s research and the co-authors have acted in guiding roles only.

2.1 Introduction

Previous studies on arrays of tidal turbines have shown that in order to correctly assess the power extraction from tidal turbine arrays, an undisturbed flow approach (also termed the flux method (BLACK & VEATCH, 2012)), where local and regional flow changes due to the presence of the turbines are not integrated into the model, does not suffice. In order to compute an accurate estimate of power production, the hydrodynamic influences of the turbines and their wake interactions must be accounted for (Garrett and Cummins, 2007; Nishino and Willden, 2012a; Vennell, 2010; Whelan et al., 2009). Therefore, a numerical model that aims to examine the power output and environmental impacts of arrays of tidal turbines must be able to capture these features.
Chapter 2: The ADM-RANS model

Currently many large-scale marine hydrodynamic models employed to study marine energy use the depth-averaged shallow water equations (Sutherland et al., 2007; Divett et al., 2013; Martin-Short et al., 2015). In order to numerically simulate arrays of turbines, these models usually adopt a discrete approach where the turbines are represented as a region of increased bottom drag and the relatively low complexity of such an approach allows for adjoint-based optimisation techniques to be used to improve turbine positions (Funke et al., 2014). More recently, Funke et al. (2016) extended this to find the optimal turbine density, i.e. the number of turbines per unit area represented as a continuous field. These models benefit from relatively low computational costs that allow for large scale realistic simulations, such as modelling arrays of tidal turbines in the Pentland Firth (Martin-Short et al., 2015). However, the main shortcoming of such an approach is that it can fail to account for important turbulence physics and 3D effects, e.g. since the flow passing below and above the turbine is not modelled. Alternatively a higher fidelity 3D model coupled with an appropriate turbulence model can be used. A number of these models have been developed and validated against experimental flume tests with promising results (Afgan et al., 2013; Roc et al., 2013), but their high computational expense has generally prevented their application in larger scale regional simulations.

Mesh optimisation techniques can help bridge the gap and improve the accuracy of large scale simulations without the need for excessive computational power. Creech et al. (2012) utilised dynamic mesh optimisation to develop a high fidelity ADM-LES model to accurately model the flow past wind turbines. Furthermore, Divett et al. (2013) developed a 2D depth-averaged model coupled with an adaptive mesh flow solver to demonstrate the greater energy extraction that can be achieved from turbines arranged in staggered layouts.

The model presented here is based on actuator disc momentum (ADM) theory, where the turbines are represented as momentum sink terms. This 3D approach is coupled with a mesh optimisation algorithm that employs a fine spatial resolution only in regions of interest. This allows for an accurate turbine wake characterisation whilst maintaining a relatively low overall computational cost. An ultimate aim will be to use this model to assess the effects of deploying arrays of tidal turbines in realistic domains, such as the Inner Sound of the Pentland Firth where MeyGen Ltd plan to deliver a fully operational 398MW renewable energy plant powered purely by the
An alternative approach to ADM theory would be to apply BEM theory, whereby radially varying forces in both axial and azimuthal directions are applied. This method better represents the performance of a real turbine as it introduces a swirl component in the wake profile. Stallard et al. (2015) investigated the mean wake properties behind a single three-bladed rotor and demonstrated that between 0.5–2 diameters downstream of the turbine, BEM theory can account for the near wake properties reasonably well. Furthermore, Batten et al. (2013) demonstrated that the swirl component of the wake dissipates quickly in the streamwise direction and, although the near wake profile produced by ADM and BEM methods are different, the far wake profiles are very similar. Given that the aim of the current study is to develop a tool for array design, as long as the turbines are not positioned in very close proximity to each other and in the near wake of upstream turbines, neglecting swirl remains a reasonable assumption. The main aim here is to achieve accurate far wake representation and correct energy yield estimates.

This chapter is organised as follows. First in section 2.2 the Fluidity ADM-RANS model is presented along with a description of the turbulence models used. The mesh optimisation techniques employed in the Fluidity model are also briefly explained. Then in section 2.3 verification against the Conway solution for flow past an actuator disc (Conway, 1995) is presented. Furthermore, a similar OpenFOAM ADM model has been developed and a comparison between this model and the Fluidity model is presented to demonstrate the benefits of the finite element discretisation used in the Fluidity model. The chapter concludes with a general overview of the results and a discussion on the benefits of the model presented.

2.2 Methodology

The numerical model presented has been developed within the Fluidity framework, an open source finite element CFD code with 3D mesh optimisation capabilities (Piggott et al., 2008). The main feature of Fluidity that motivates this study is its ability to adapt the mesh dynamically in time and only in the locations of interest. The code is also highly parallelisable which makes it attractive for larger scale com-
Chapter 2: The ADM-RANS model

putationally challenging applications.

2.2.1 Governing equations

The importance of accounting for the ambient turbulence when modelling the flow past tidal turbines was discussed in section 1.4.2. For the purpose of this study it has been decided to incorporate turbulence models based upon the Reynolds-averaged Navier-Stokes (RANS) approach, given that their computational cost is significantly lower than that required with large eddy simulations (LES). These models are based on the RANS equations, in which the velocity is decomposed into mean and fluctuating (turbulent) components

$$\rho \frac{\partial \mathbf{u}}{\partial t} + \rho \mathbf{u} \cdot \nabla \mathbf{u} = -\nabla p + \mu \nabla^2 \mathbf{u} - \nabla \cdot (\rho \mathbf{u}' \otimes \mathbf{u}') + S_u,$$

(2.1)

where $\otimes$ denotes the outer product, $\mathbf{u}$ is the mean velocity, $\mathbf{u}'$ is the fluctuating velocity, $\rho$ is the fluid density, $p$ is the mean pressure, $\mu$ is dynamic viscosity and $S_u$ is the momentum sink term included here to account for the presence of the turbine. The third term on the right hand side, containing the Reynolds stress tensor $-\rho \mathbf{u}' \otimes \mathbf{u}'$, represents the effect of turbulent fluctuations on the mean flow and for incompressible flows is defined as

$$- \rho \mathbf{u}' \otimes \mathbf{u}' = \tau_R = -\frac{2}{3} k \rho I + \mu_T \left( \nabla \mathbf{u} + (\nabla \mathbf{u})^T \right),$$

(2.2)

where $k = (\mathbf{u}' \cdot \mathbf{u}')/2$ is the turbulent kinetic energy and $\mu_T$ is the dynamic eddy viscosity.

The 3D Fluidity model presented uses a P1DG – P2 finite element pair to discretise the RANS equations. This scheme uses the space of discontinuous piecewise linear functions (P1DG) to represent velocity and the space of continuous piecewise quadratic functions (P2) for pressure. This is a stable element pair on tetrahedra which has benefits in geophysical applications with Coriolis forces (Cotter et al., 2009) that become important at the larger scale where the turbine simulations will be embedded in the future.
The Fluidity model uses a Crank–Nicolson time discretisation which is second-order accurate in time. However, in order to achieve a stable and bounded solution with the discontinuous Galerkin (DG) discretisation, a slope limiter is used in conjunction with an explicit treatment of the advection term. Therefore, the RANS equations are considered in two stages where initially advection is considered, and then the other terms. For advection, explicit subcycles utilising adaptive sub-timesteps are used in order to satisfy a specified (sub-unity) Courant–Friedrichs–Lewy (CFL) condition. A larger overall timestep and Crank–Nicolson discretisation are used for the remaining terms including diffusion, pressure gradient and sources. For further details, the reader is referred to Parkinson et al. (2014) and the Fluidity manual (Applied Modelling and Computation Group (AMCG), 2015).

### 2.2.2 Actuator Disc Momentum theory

The 3D numerical model developed and validated here incorporates turbines which are parametrised based on the ADM theory outlined by Houlsby et al. (2008). ADM theory is based on the assumptions that the flow is inviscid and incompressible with uniform inflow. The disc is infinitely thin and the thrust loading on the disc is uniformly spread. Given that many of these assumptions do not hold in the numerical model, or the real world, it is vital to verify the implementation and to validate the outcome from the model against laboratory and real world data.

In the current model the circular disc representing the turbine has a small finite thickness and is represented as a scalar turbine field which takes the value unity at the location of the disc and is zero everywhere else in the domain. This scalar field is discretised using the same space as the velocity field; i.e. P1DG. This ensures that the disc is present only in the intended region and does not spread into the surrounding elements, which would be the case if a continuous Galerkin (CG) discretisation was used. In the Fluidity model an unstructured mesh is used to capture the circular actuator disc accurately and the disc is given a unique mesh region ID, which is important in the context of the mesh optimisation techniques used (section 2.2.5).

In order to set the appropriate loading on the disc, the Fluidity model uses the established definition for thrust coefficient, $C_t$, to compute the magnitude of thrust.
loading that should be applied at the disc. This is uniformly spread across the volume of the disc and is implemented as a momentum sink term

\[ S_u = -\frac{1}{V_{\text{disc}}} \left( \frac{1}{2} \rho A_{\text{disc}} C_t u_0^2 \right), \]  

(2.3)

where \( S_u \) is the momentum sink applied only at the location of the disc, \( A_{\text{disc}} \) is the cross-sectional area of the disc, \( V_{\text{disc}} \) is the volume of the disc and \( u_0 \) is the unperturbed upstream streamwise component of velocity. In the context of the numerical simulations, it is important to ensure that \( u_0 \) is predicted accurately; this being discussed in detail as part of the model validation in chapters 3 and 4.

### 2.2.3 The \( k - \omega \) model

One of the most widely used RANS models is the \( k - \omega \) model where the momentum equations are closed by solving transport equations for \( k \) and the turbulent frequency, \( \omega \),

\[
\rho \frac{\partial k}{\partial t} + \rho \mathbf{u} \cdot \nabla k = \nabla \cdot \left( (\mu + \sigma_k \mu_T) \nabla k \right) + P_k - \rho \beta^* k \omega + S_k, \tag{2.4}
\]

\[
\rho \frac{\partial \omega}{\partial t} + \rho \mathbf{u} \cdot \nabla \omega = \nabla \cdot \left( (\mu + \sigma_\omega \mu_T) \nabla \omega \right) + \left( \frac{\rho \alpha}{\mu_T} \right) P_k - \rho \beta \omega^2 + S_\omega, \tag{2.5}
\]

where \( \mathbf{u} \) is the mean velocity (with the overbar dropped for convenience from here onwards), the dynamic eddy viscosity is defined as

\[
\mu_T = \rho k / \omega,
\]

and

\[
P_k = \bar{T}_R \cdot \nabla \mathbf{u},
\]

is the turbulence production term in the model. The other coefficients in (2.4) and (2.5) take the values \( \sigma_k = 0.5, \sigma_\omega = 0.5, \beta = 0.075, \beta^* = 0.09 \) and \( \alpha = 5/9 \) (Wilcox,
2.2: Methodology

$S_k$ and $S_\omega$ represent additional source terms which are not present in the original $k-\omega$ model, but are included here to account for the presence of the turbine as described below.

Alternatively, the $k-\varepsilon$ model is a popular approach which can be used where the momentum equations are closed by solving transport equations for $k$ and the turbulent dissipation, $\varepsilon$, where $\varepsilon = \beta^* \omega k$. However, both of the $k-\omega$ and $k-\varepsilon$ models tend to predict faster wake recoveries in comparison with experimental data, as demonstrated by Roc et al. (2013). The reason for this lies in the fact that they fail to account for the correct energy transfer rate from large-scale turbulence to small-scale turbulence in the near-wake region. This is known as the short circuiting of the turbulence cascade due to the presence of the turbine (El Kasmi and Masson, 2008).

To overcome this, turbulence correction terms have been suggested in both the tidal and wind turbine literature (El Kasmi and Masson, 2008; Rethore et al., 2009; Roc et al., 2013). Rethore et al. (2009) and Roc et al. (2013), include additional terms for the turbulent kinetic energy (TKE) equation (2.4). They suggest including an additional source term $S_{k,p}$ which accounts for the production of wake turbulence and also include an additional sink term $S_{k,d}$ to account for the short circuiting of the turbulence cascade. In this study the approach by Rethore et al. (2009) has been followed where the source and sink terms are scaled with $C_t$ and applied only at the disc. These are calculated via

$$S_k = S_{k,p} - S_{k,d} = \frac{1}{2} C_x (\beta_p |u|^3 - \beta_d |u| \cdot k),$$

with

$$C_x = \frac{4a}{1 - a},$$

and

$$a = \frac{1}{2} \left(1 - \sqrt{1 - C_t}\right),$$

where $a$ is the axial induction factor (Hansen, 2000), $\beta_p = 0.05$ and $\beta_d = 1.5$ (Rethore et al., 2009). Note that the Eq. (2.8) is only applicable when applying linear momentum theory in unconstrained flow and therefore requires negligible
local blockage. El Kasmi and Masson (2008) suggest an additional production term for the $k - \varepsilon$ model which is proportional to the quadratic production of TKE by shear and is applied at the disc $\pm 0.25$ diameters directly upstream and downstream of the disc. Rados et al. (2009) extended this for the original $k - \omega$ model and compute an additional source term, $S_\omega$, via

$$S_\omega = C_\omega \frac{1}{\rho k^2} (P_k)^2,$$

(2.9)

with $C_\omega = 4$ (Rados et al., 2009). This is the definition of $S_\omega$ used in this study.

To sum up, in this study $S_{k,d}$ and $S_\omega$ are included to capture the short-circuiting of the turbulence cascade at the turbine and $S_{k,p}$ is included to represent the additional production of wake turbulence.

### 2.2.4 The $k - \omega$ SST model

It has been suggested that a $k - \omega$ model is better suited for modelling separating flows compared to the $k - \varepsilon$ model due to the latter’s inability to capture turbulence correctly in near-wall regions (Wilcox, 1998). However, despite the $k - \omega$ model’s superior performance in the near wall region, its strong sensitivity to the freestream $\omega$ values has prevented it from becoming the standard model of choice for RANS modelling (Menter, 1992). This shortcoming was one of the main motivations for the development of the $k - \omega$ shear stress transport (SST) model by Menter (1994) whereby blending functions are introduced so that the $k - \omega$ model is used in the inner region of the boundary layer and the $k - \varepsilon$ model is used in the outer region and in free shear flows. Furthermore, the definition of the eddy viscosity is modified to account for the effect of the transport of the principal turbulent shear stress (Menter, 1994). In the $k - \omega$ SST model the transport equations are modified to

$$\rho \frac{\partial k}{\partial t} + \rho \mathbf{u} \cdot \nabla k = \nabla \cdot (\mu_T \nabla k) + \tilde{P}_k - \rho \beta^* k \omega,$$

(2.10)

$$\rho \frac{\partial \omega}{\partial t} + \rho \mathbf{u} \cdot \nabla \omega = \nabla \cdot ((\mu + \sigma \mu_T) \nabla \omega) + \left(\frac{\rho \alpha}{\mu_T}\right) \tilde{P}_k - \rho \beta \omega^2 + 2(1 - F_1) \rho \sigma \omega \frac{1}{\omega} \nabla k \cdot \nabla \omega,$$

(2.11)
where the blending function $F_1$ is used to switch from the $k-\omega$ model, when $F_1 = 1$, to a $k-\varepsilon$ model, when $F_1 = 0$ and the limiting

$$\tilde{P}_k = \min(P_k, 10\rho\beta^*k\omega),$$

is applied in order to prevent the build-up of turbulence in stagnation regions (Menter et al., 2003). Furthermore, all the coefficients in the transport equations (2.10) and (2.11), are computed using a blend from the coefficients of the original $k-\omega$ and $k-\varepsilon$ RANS models. For example

$$\sigma_k = \sigma_{k1}F_1 + \sigma_{k2}(1 - F_1),$$

with $\sigma_{k1} = 0.85$, $\sigma_{k2} = 1$, $\sigma_{\omega1} = 0.5$, $\sigma_{\omega2} = 0.856$, $\alpha_1 = 5/9$, $\alpha_2 = 0.44$, $\beta_1 = 0.075$, $\beta_2 = 0.0828$ and $\beta^* = 0.09$ (Menter et al., 2003). The $k-\omega$ SST model implemented in Fluidity is based on the one presented in Menter et al. (2003) with the $F_1$ blending function defined as

$$F_1 = \tanh \left( \min \left( \max \left( \frac{\sqrt{k}}{\beta^*\omega y}, \frac{500\nu}{y^2\omega}, \frac{4\rho\sigma_{\omega2}k}{CD_{k\omega}y^2} \right) \right) \right)^4,$$

where $\nu$ is the background kinematic viscosity, $y$ is the dimensional distance to the nearest wall and

$$CD_{k\omega} = \max \left( 2\rho\sigma_{\omega2} \frac{1}{\omega} \nabla k \cdot \nabla \omega, 10^{-10} \right).$$

Also, the definition for the dynamic eddy viscosity used is

$$\mu_T = \frac{\rho a_1 k}{\max(a_1\omega, SF_2)};$$

where $a_1 = 0.31$ and $S$ is the so called strain invariant.
\[
S = \sqrt{2}(S \cdot S),
\]
\[
S = \frac{1}{2} \left( \nabla u + (\nabla u)^T \right),
\]

with the second blending function, \( F_2 \), defined as
\[
F_2 = \tanh \left( \max \left( \frac{2\sqrt{k}}{\beta^*\omega y}, \frac{500\nu}{y^2\omega} \right)^2 \right).
\]

### 2.2.5 Mesh optimisation

The Fluidity ADM model is capable of dynamic mesh optimisation which is used to help reduce discretisation errors by refining the mesh in locations of numerical complexity or specific interest, e.g. regions with high velocity shear which would be valuable in accurate wake representation. This is achieved via the construction of a metric tensor field based upon the Hessians of solution fields, user-defined error bounds, and maximum and minimum allowed edge lengths \( l_e \). Thereafter, using this metric, a functional of the mesh elements is formed whose minimisation through heuristic topological operations ensures a mesh with elements which are appropriately shaped and sized. Consequently, all the field data from the previous mesh is projected onto the new mesh by interpolation. As a result of this the mesh is not only refined in regions of interest, it is also coarsened in regions where high resolution is not required. This helps reduce the computational cost over a tidal simulation where the regions of interest within the domain continuously change. In this study consistent interpolation has been used in Fluidity where the data from the previous mesh is transferred onto the new mesh via an evaluation of the basis functions.

In the interest of brevity, a detailed description of the mesh optimisation techniques is omitted here, and reference is made to the extensive work by Piggott et al. (2008, 2009) and Pain et al. (2001). For the purpose of this work the mesh is refined in regions of high curvature in the velocity, \( k \) and \( \omega \) fields, motivated by the desire to correctly capture the re-energisation of the wake downstream of the turbines. While mesh optimisation techniques by Piggott et al. (2008, 2009) and Pain et al. (2001)
are now well established, they have never been demonstrated with the turbulence models described under sections 2.2.3–2.2.4. Indeed, the combined application of accurate turbulence modelling and efficient mesh optimisation forms one of the key advances of the present work, and is central to the description of the flow fields in the wake of tidal turbines.

Furthermore, as previously mentioned, the actuator disc is given a different mesh region ID to the rest of the domain. In the Fluidity simulations, the mesh region IDs are preserved and therefore the boundary between the actuator disc and the rest of the domain is protected and does not change during mesh optimisation. Hence, the volume of the actuator disc does not change as a result of the mesh optimisation techniques used. This is important since a change in volume needs to be avoided given that it will affect the magnitude of the momentum sink term applied (Eq. (2.3)) in the ADM-RANS model.

2.3 ADM verification

The implementation of the ADM model in Fluidity is new and must be verified. In this section this implementation is assessed against a semi-analytical solution and a similar ADM model developed in OpenFOAM. Note that, the RANS turbulence models and the mesh optimisation techniques are not investigated in this section. These will be discussed in great detail in chapters 3–4.

2.3.1 Semi-analytical solution

Conway (1995) suggests a semi-analytical form for the velocity profile of the wake behind an actuator disc and this will be used to help verify the numerical ADM implementations; the approach adopted being similar to that by Viré et al. (2013). Assuming incompressible and inviscid flow, the velocity profile takes the form

\[ u_x(x, r) = u_0 - \frac{\nu_{\text{wake}}}{4} \left( \frac{x}{\pi \sqrt{rD/2} Q_{-1/2}(w)} + \Lambda_0(b, h) + 2 \right), \quad (2.12) \]

if \( r < D/2 \) and
\[ u_x(x, r) = u_0 - \frac{u_{\text{wake}}}{4} \left( \frac{x}{\pi \sqrt{rD/2}} Q_{-1/2}(w) - \Lambda_0(b, h) \right), \]  
(2.13)

if \( r > D/2 \),

where \( u_x \) is the velocity in the streamwise direction, \( u_0 \) is the unperturbed upstream velocity in the streamwise direction, \( Q_{-1/2}(w) \) denotes the Legendre function of the second kind, \( \Lambda_0(b, h) \) denotes Heuman’s Lambda function, \( x \) is the streamwise direction, \( r \) is the axial direction and \( D \) is the disc diameter. The only unknown in the solution is \( u_{\text{wake}} \), the streamwise component of velocity in the wake which is sufficiently far downstream from the turbine that the pressure can again be treated as uniform. In this study \( u_{\text{wake}} \) has been computed using Houlsby et al. (2008):

\[ \frac{u_{\text{wake}}}{u_0} = \sqrt{1 - Ct}. \]  
(2.14)

Similar to Eq. (2.8), linear momentum theory in unconstrained flow is applied here and therefore negligible local blockage is assumed. The other parameters are computed from \( x, r \) and \( D \) as follows:

\[ w = \left( x^2 + r^2 + D^2/4 \right)/rD, \quad h = \sqrt{2rD/m_1}, \quad b = \arcsin(x/\sqrt{m_2}), \]
\[ m_1 = x^2 + (D/2 + r)^2, \quad m_2 = x^2 + (D/2 - r)^2. \]

### 2.3.2 OpenFOAM ADM model

To provide an additional means of comparison, the same ADM implementation described above has also been applied to the PISO (Pressure Implicit with Splitting of Operators) solver in OpenFOAM, an open source finite volume CFD package (OpenFOAM, 2015). This was achieved following a similar approach to that described in Svenning (2010). OpenFOAM is a well-established CFD package and the OpenFOAM ADM model will help verify the implementation in Fluidity. It will also aid in identifying the potential benefits of the finite element approach used in Fluidity. As with the Fluidity model, the disc has a small finite thickness and is represented as a scalar field which is unity at the location of the disc and zero everywhere else in the domain, with the momentum sink term, (2.3), applied at the disc
only. Furthermore, the cell-centred finite volume discretisation used in OpenFOAM ensures that the turbine scalar field is only present where intended, similar to the P1DG finite element discretisation used in the Fluidity model described in section 2.2.2.

### 2.3.3 Results

![Front View](image1.png) ![Side View](image2.png)

**Figure 2.1:** A cut through the tetrahedral Fluidity mesh on the left and the hexahedral OpenFOAM mesh on the right with the disc positioned at the centre. The meshes have been regularly extruded in the streamwise direction with a layer thickness of 1m which is equal to the disc thickness. Note that a finer mesh resolution has been used close to the disc with minimum \(l_e/D = 0.10\) in the Fluidity mesh and minimum \(l_e/D = 0.02\) in the OpenFOAM mesh.

**Figure 2.2:** Numerical domain with \(D = 20\) m used in both Fluidity and OpenFOAM simulations corresponding to the comparison plots presented in Fig. 2.3–2.4. The dark grey area represents the actuator disc.

In order to computationally reproduce the inviscid flow assumption of the Conway
solution, a low value for kinematic viscosity was used in both Fluidity and OpenFOAM, \( \nu = 1 \times 10^{-6} \text{ m}^2 \text{s}^{-1} \). The two numerical models have used different, yet similar size meshes where a tetrahedral mesh was used for Fluidity and a hexahedral mesh was used for OpenFOAM. The domain extent and the disc size were kept the same with a 0.25% difference in disc volume between the two meshes. The total number of elements was also very similar with the Fluidity mesh containing \( 8.47 \times 10^5 \)
2.3: ADM verification

Figure 2.4: Radial comparison velocity deficit plots with \( Re = 6.0 \times 10^7 \) at \( C_t = 0.2 \) and \( C_t = 0.8 \). The same legend as in Fig. 2.3 is used.

A timestep of 0.1 s was used in OpenFOAM resulting in a maximum Courant number of \( \sim 0.38 \). In the Fluidity simulations an overall timestep of 1 s was used leading
Chapter 2: The ADM-RANS model

to a maximum overall Courant number of $\sim 11$. However, adaptive sub-timesteps were chosen to ensure a maximum Courant number of $\sim 0.1$ for the advection term, as described in section 2.2.1, which ensured numerical convergence.

Fig. 2.3–2.4 show velocity deficit plots comparing the numerical models to the semi-analytical solution for $C_t = 0.2$ and $C_t = 0.8$ with the numerical domain illustrated in Fig. 2.2. The velocity deficits are well predicted by both the Fluidity and the OpenFOAM models and the plots demonstrate that using a higher $C_t$ leads to a greater velocity deficit, as expected. Along $r = 0$ the Fluidity model provides a better match to the semi-analytical solution and a closer look at Fig. 2.3a reveals that the Fluidity model performs better at capturing the sharp velocity variation present at the disc location, whereas the OpenFOAM model exhibits some fluctuations. Along $r = D$ the simulated results diverge from the semi-analytical solution and this is due to the infinite domain assumed in the Conway solution, whereas the wall effect in the numerical simulations causes the velocity outside of the wake, in the bypass region, to accelerate. This effect can be reduced by moving the walls further away from the disc; i.e. using a larger computational domain and a smaller blockage ratio. This is demonstrated in Fig. 2.3b where a larger numerical domain has been used. Increasing the domain size indeed leads to an improved agreement due to reduced blockage where the local blockage, Eq. (1.1), has been reduced from 0.78% to 0.20%. The remaining departures are of order $< 0.01 u_0$, which is considered acceptable.

In the radial direction, although both models struggle to provide a close match to the Conway solution just behind the disc, Fig. 2.4a, these discrepancies are drastically reduced two diameters downstream, Fig. 2.4b, and both models follow the analytical solution with the Fluidity results providing a closer match. Also, note that in Fig. 2.4a the Fluidity model slightly overshoots, indicating that the bypass flow is accelerating and therefore exceeds $u_0$. However, this is not observed in the OpenFOAM model possibly due to the mesh resolution used resulting in a more dissipative velocity profile. Furthermore, no acceleration is visible in the bypass flow in the semi-analytical solution since an infinite domain is assumed.

Overall, considering the assumptions used in the semi-analytical solution, both numerical models simulate the inviscid flow past the actuator disc accurately. More-
over, the $P_{1DG} - P_2$ finite element discretisation employed in the Fluidity model has helped in providing a closer match to the Conway solution compared to the OpenFOAM model.

2.4 Conclusions

An ADM model incorporating a momentum sink term and RANS models which can simulate the flow past horizontal axis tidal turbines and account for turbulence characteristics has been developed within the Fluidity adaptive mesh CFD solver. The ADM implementation has been verified against the Conway semi-analytical solution. Furthermore, a comparison against an OpenFOAM model based on the same ADM methodology has also been presented to portray the benefits of the $P_{1DG} - P_2$ finite element discretisation scheme employed in the Fluidity ADM-RANS model. In the following chapters the suitability of the RANS models implemented will be assessed. Moreover, the mesh optimisation techniques available in the Fluidity ADM-RANS model will be utilised to obtain an accurate representation of the tidal turbine wakes whilst maintaining relatively low computational costs.
Chapter

THREE

ADM-RANS validation: Flow past a porous disc

This chapter is derived from and significantly expands upon a conference paper published in the Proceedings of the 11th European Wave and Tidal Energy Conference by Abolghasemi et al. (2015). The material therein has arisen as the result of the first author’s research and the co-authors have acted in guiding roles only.

3.1 Introduction

The real world environment where tidal turbines will be installed contains complex flow patterns and a better understanding of the flow regime is vital prior to the deployment of any turbines. Numerical simulations are a valuable tool which can help expand our knowledge of tidal flows. However, due to the complexity of the problem, assumptions about the flow have to be made in order to tackle the problem in line with the computational power available. Undoubtedly, given the technological advances in the past decade in the field of high performance computing, improved computational power will enable less crude approximations of the flow regime, but even then the benefits of a more detailed solution will have to be balanced against the cost of additional computational power. Hence, it is crucial to understand the
effects of the approximations involved in the numerical solutions and the physical phenomena that might be misrepresented as a result of the assumptions made.

Experimental analyses of the flow past tidal turbines are aimed at improving our understanding of the flow regime and its interaction with the turbines therein. Only a handful of tidal turbines have been deployed in a real tidal environment up until now. In 2008, the SeaGen S 1.2MW device located in Strangford Lough was the world’s first grid connected commercial scale tidal device, but despite its success, very few commercial scale projects have been completed since then. Another notable example is the DEEP-Gen IV 1MW turbine installed in 2013 at the European Marine Energy Centre (EMEC) site as part of the Energy Technology Institutes (ETI) Reliable Data Acquisition Platform for Tidal (ReDAPT) project (ALSTOM, 2015). As a result of this and given the difficulties associated with taking measurements in these turbulent locations, as well as confidentiality concerns, there is a lack of experimental data regarding the flow past tidal turbines in a real environment.

A wide range of experimental studies using scaled turbines (Mycek et al., 2014a,b; Stallard et al., 2013) and porous discs (Blackmore et al., 2014; Myers and Bahaj, 2010) have been conducted, all aimed at recreating the real life scenario on a laboratory scale. However, at the start of this PhD there were limited experimental datasets available, largely focused on turbine loading rather than wake structure (e.g. Bahaj et al. (2007)). Hence, a series of experiments were planned in order to help build our understanding of the flow past tidal turbines and to guide, inspire and motivate the development of the numerical model. These experiments used porous disc rotor simulators and were carried out in the 4ft current flume of the hydrodynamics laboratory. The 4ft current flume is 8 m long, 1.2 m wide with a depth of up to 1.1 m and can support flow rates up to 240 litres per second. Following the experiments, the Fluidity ADM-RANS model, described in chapter 2, is used to simulate the experimental scenario. This allows for a thorough assessment which will help highlight the limitations of the numerical model.

This chapter is organised as follows. First in section 3.2 the scaling used is explained and the apparatus used in the experiments is described in section 3.3. The ambient flume conditions are presented in section 3.4. The porous disc thrust coefficients are determined in sections 3.5 and section 3.6 presents the results from the wake
profile measurements. Thereafter, in section 3.7 the laboratory scenario is replicated numerically via the ADM-RANS model and the suitability of the RANS models employed and mesh optimisation techniques used is assessed in detail. The chapter concludes with a general overview of the results.

3.2 Similitude

Ensuring dynamic similitude is vital when attempting to recreate a real life scenario in a laboratory (Hughes, 1993) and Froude similarity must be established when considering tidal turbines to ensure that the free surface at the model scale deforms in the same manner as the full-scale system (Myers and Bahaj, 2010). With this in mind and given the dimensions of the 4ft flume, it was decided to conduct the experiments at a scale of 1:125. Table 3.1 shows how this scaling affects the other important parameters. The Reynolds numbers \( Re \) have been calculated based on the water depth as the characteristic length.

<table>
<thead>
<tr>
<th></th>
<th>turbine diameter</th>
<th>water velocity</th>
<th>water depth</th>
<th>hub height</th>
<th>( Fr )</th>
<th>( Re )</th>
</tr>
</thead>
<tbody>
<tr>
<td>physical</td>
<td>18.75 m</td>
<td>2.90 m s(^{-1})</td>
<td>33 m</td>
<td>16 m</td>
<td>0.161</td>
<td>( 9.6 \times 10^7 )</td>
</tr>
<tr>
<td>experiment</td>
<td>0.15 m</td>
<td>0.26 m s(^{-1})</td>
<td>0.264 m</td>
<td>0.13 m</td>
<td>0.161</td>
<td>( 6.9 \times 10^4 )</td>
</tr>
</tbody>
</table>

**Table 3.1:** Experimental scale of the 4ft flume experiments where a 1:125 scale is used.

An important issue to note is the difference between the full scale and the laboratory scale Reynolds numbers. Attempting to recreate a laboratory setting where the \( Re \) would match is almost impossible; therefore it was decided to conduct the experiments at a sufficiently high \( Re \) such that the flow is in the turbulent regime and the value of \( 6.9 \times 10^4 \) (Table 3.1) satisfies this criteria (Pope, 2000).
3.3 Methodology

The experiments consisted of measuring the wake profile downstream of a porous disc rotor simulator placed inside the 4ft flume, Fig. 3.1. The inlet velocity was set to $(0.26 \pm 0.01)$ m s\(^{-1}\) and the water depth was fixed to $(0.27 \pm 0.01)$ m. In this section the apparatus used in the experiments is described in detail.

![A photograph of the porous disc inside the 4ft flume.](image)

**Figure 3.1:** A photograph of the porous disc inside the 4ft flume.

3.3.1 Porous discs

At a scale of 1:125 it is no longer practical to accurately scale both thrust and tip speed without altering downstream flow properties (Myers and Bahaj, 2010). For example, if it is assumed that a real turbine with a diameter of 18.75 m rotates at 20 rpm at peak inflow velocity, this would correspond to a tip speed of 19.6 m s\(^{-1}\). For the scaled turbine to achieve the same tip speed, it will have to rotate at 2500 rpm. This is impractical from a design point of view and it is also undesirable since it will introduce a significant amount of swirl into the flow and will create large pressure gradients. It is believed that reproducing the thrust accurately is of primary importance. Therefore, the tidal turbines are represented using porous discs which will reproduce the thrust exerted on the rotor. The magnitude of the thrust can be controlled by varying the level of porosity of the disc.

Using porous discs to represent rotors will undoubtedly lead to different downstream
flow properties. There will be no swirl induced onto the flow from the porous disc and the energy extracted via the disc is converted into small scale turbulence as opposed to being extracted as mechanical motion. Furthermore, the vortices shed from the edge of the disc will differ from those of a rotating blade. However, none of these differences are expected to effect the far field flow properties in the wake and are only expected to influence flow properties in the near wake region (Myers and Bahaj, 2010).

The porous discs used in this study have been constructed out of perforated sheets of aluminium. Table 3.2 describes the properties for the different discs considered. Note that the thickness values are not the same due to the perforated aluminium sheets available for the desired porosities, but in both cases the thickness values are negligible compared to the disc diameters. Although, the thrust applied on the flow by the disc is considered to be primarily dependent on the porosity of the disc, hole diameter is also believed to have an effect. Myers and Bahaj (2010) suggest that due to orifice type losses associated with reducing hole diameter, for discs with equal porosity, smaller and more numerous holes will result in a higher disc thrust.

<table>
<thead>
<tr>
<th>diameter (m)</th>
<th>thickness (mm)</th>
<th>hole diameter (mm)</th>
<th>porosity (%)</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.15</td>
<td>0.6</td>
<td>1.7</td>
<td>45</td>
</tr>
<tr>
<td>0.15</td>
<td>1.2</td>
<td>4.8</td>
<td>50</td>
</tr>
</tbody>
</table>

*Table 3.2: Porous discs properties.*

### 3.3.2 Acoustic doppler velocimeter (ADV)

In order to capture the velocity profile downstream of the porous disc, the methodology employed here follows similar approaches by Myers and Bahaj (2010) and Stallard et al. (2013) where a Nortek Vectrino acoustic doppler velocimeter (ADV) was used. The ADV is capable of recording instantaneous velocity components to within ±1%, in three directions, at a single point. A Dantec Dynamics modular traverse system was used to translate the ADV and map out the velocity profile downstream of the disc. At each location, the ADV recorded the velocity for 60s and by taking advantage of the ADV’s high frequency sampling rate of 200 Hz,
experimental turbulence kinetic energy \( k_{\text{exp}} \) and turbulence intensity \( I_{\text{exp}} \) values have also been diagnosed from the measured velocities. By definition, the turbulence kinetic energy is given by

\[
k_{\text{exp}} = \frac{1}{2} \left( (u'_x)^2 + (u'_y)^2 + (u'_z)^2 \right),
\]

(3.1)

where \( u'_x \) is the instantaneous turbulence velocity fluctuations in the \( x \) direction and similarly \( u'_y \) and \( u'_z \) are the components in the \( y \) and \( z \) directions respectively. The experimental turbulence intensity is given by

\[
I_{\text{exp}} = \sqrt{\frac{2}{3} k_{\text{exp}}} \frac{U}{U},
\]

(3.2)

where \( U \) is the magnitude of the inlet flume velocity.

In addition to the velocity readings, the Vectrino ADV provides a correlation reading for every measurement. The correlation coefficient \( (R^2) \) is a measure of signal quality and is computed using

\[
R^2 = e^{-2\pi^2 \phi^2},
\]

(3.3)

where \( \phi^2 \) is the dimensionless spectral width (Myers and Bahaj, 2010). Nortek recommend a value greater than 70% for turbulent flows and therefore all data points that did not meet this criterion have been removed. This helps remove noise associated with the ADV signal and removes spurious velocity spikes. Furthermore, any data points that did not lie within three standard deviations of the mean velocity are also disregarded in order to remove any remaining velocity spikes, if present.

\subsection*{3.3.3 Load cell}

The axial loading on the disc has to be accurately measured in order to determine its corresponding \( C_t \). The load cell used was a 2lb (8.9N) FUTEK Miniature S Beam load cell and in order to ensure the correct loading is recorded, the disc was free to rotate about the pivot point where its motion was restricted via the load cell,
3.4: Ambient flume conditions

Fig. 3.2. The distance from the pivot to the load cell was the same as the distance from the centre of the disc to the pivot. The data acquisition system was set to sample the load cell signal at a frequency of 1000 Hz with a sample period of 60 s and the mean value was chosen.

Figure 3.2: Axial loading on the disc was recorded using a pivoting system.

Capturing the loading on the discs accurately is very important since it is used to calibrate the associated disc thrust coefficient and therefore errors in these measurements should be minimised as far as possible. One of the issues regarding the current setup is that the pivot used is assumed to be perfectly frictionless. This is clearly not the case and the setup might underestimate the loading on the disc. Another more important issue is that the disc is not free to rotate at the hub since it is rigidly attached to the rod. Hence, bending moments experienced by the disc will cause the disc-rod structure to rotate about the pivot point and will therefore manifest themselves as a reading on the load cell. Therefore, the recorded measurements might correspond to more than just the axial loading on the disc.
Chapter 3: ADM-RANS validation: Flow past a porous disc

Figure 3.3: Boundary layer profile: The water depth is approximately 270 mm but measurements only exist up to 215 mm since the ADV measures the velocity at a point 50 mm below its position and only operates in water. The three profiles shown correspond to one recorded in the centre of the flume and two measurements taken 0.3 m either side of the centre.

3.4 Ambient flume conditions

Prior to conducting experiments in the flume it is important to understand the experimental flow conditions. Hence, the ADV was used to measure the flow profile of the flume without the presence of any discs. Fig. 3.3 shows the streamwise velocity variations with depth measured at three different locations in the tank. One in the centre and two measurements taken 0.3 m either side of the centre which is also 0.3 m away from the side walls, since the flume width is 1.2 m. It is therefore no surprise to see that the off-centre profiles differ from the one measured at the centre, most likely due to the shear introduced at the side walls. The following log-law can be used to fit a boundary layer profile to the data through finding $z_0$

$$u = \frac{u^*}{\kappa} \ln \left( \frac{z}{z_0} \right), \quad u^* = \frac{\kappa u_{ref}}{\ln \left( \frac{z_{ref}}{z_0} \right)}$$

(3.4)

where $\kappa = 0.41$ and $u_{ref}$ and $z_{ref}$ are taken as the velocity and depth at hub height, respectively. By setting $z_0 = 1 \times 10^{-5}$ m, the log-law matches the lower regions of the flow, but does not provide a good match for the entire flow profile. Furthermore, there is a noticeable peak in the turbulence intensity values at around 5–10 mm which is due to the development of the boundary layer at the bottom surface of the flume. In the off-centre 2 profile another peak appears around 20 mm, but this
3.5: Thrust coefficient

does not appear in the other two profiles and is possibly due to a particular feature at that location in the flume bed or a spurious data point. The TKE measured inside the 4ft flume corresponds to an ambient turbulence intensity of around 5% (outside the boundary layer). This is lower than the ambient turbulence intensity in a typical ocean tidal flow which is expected to be at least 10% (Thomson et al., 2012).

3.5 Thrust coefficient

For sufficiently high Reynolds numbers, the drag/thrust ($T$) of an object is quadratically proportional to the upstream velocity $u_0$. Therefore, for the disc used in the experiments $T_{\text{disc}} \propto u_0^2$. This relationship gives rise to the well known definition for thrust coefficient ($C_t$) which is defined as

$$C_t = \frac{T_{\text{disc}}}{\frac{1}{2} \rho A_{\text{disc}} u_0^2},$$  \hspace{1cm} (3.5)

where $\rho$ is the flume water density and $A_{\text{disc}}$ is the cross-sectional area of the disc. Therefore, in order to determine the $C_t$ of the porous disc, the axial loading on the disc was recorded via the load cell for a number of different upstream velocity values while maintaining the same water depth as described above, Table 3.1. The upstream velocity was recorded via the ADV which was positioned approximately 5 m upstream of the disc recording the velocity at hub height.

Prior to determining the $C_t$ of the disc, the drag of the 10 mm circular rod which holds the porous disc in place also needs to be accounted for. Hence, the experiments were initially carried out without the disc and with only the rod in the flume. This allowed the drag coefficient of the rod to be calculated so that the rod contribution to the axial loading measurements could be accounted for. Fig. 3.4 displays the relationship between the recorded thrust reading and the upstream velocity for the rod only case. Similar to Eq. (3.5), for the rod

$$C_r = \frac{T_{\text{rod}}}{\frac{1}{2} \rho A_{\text{rod}} u_0^2},$$  \hspace{1cm} (3.6)

where $A_{\text{rod}}$ is the cross-sectional area of the portion of the rod immersed in water.
Here, $A_{\text{rod}}$ is taken as the full length of the tower (surface to hub depth).

$$A_{\text{rod}} = 0.01 \text{ m} \times 0.14 \text{ m} = 1.4 \times 10^{-3} \text{ m}^2.$$ 

Alternatively, $A'_{\text{rod}}$ can be taken as the length from the surface to the top of the disc exposed to the bypass flow plus the length of the tower (equal to the radius) exposed to the flow through the porous disc multiplied by the porosity of the disc.

$$A'_{\text{rod}} = 0.01 \text{ m} \times 0.065 \text{ m} + 0.01 \text{ m} \times 0.075 \text{ m} \times 0.45 = 9.9 \times 10^{-4} \text{ m}^2.$$ 

A quadratic fit was applied to the $T_{\text{rod}}$ vs $u_0$ data and it was determined that for the rod $C_r = 1.0$ and $C'_r = 1.4$, Fig. 3.4. A value of $C_r = 1.0$ is more in-line with the expected value for the drag of a cylinder at this Reynolds number (Anderson Jr, 2010) and therefore this is the value taken forward.

![Figure 3.4: Thrust and drag coefficients for the rod and both porous discs. The quadratic fits suggests $C_r = 1.0$ for the rod and that $C_t = 1.08$ and $C_t = 0.91$ for the 45% porous and the 50% porous discs, respectively. The error bars indicate the standard deviation associated with the load cell readings.](image)

Having established the axial loading contribution due to the rod, the $C_t$ for the porous discs described above in Table 3.2 was determined. The variation of axial
loading against upstream velocity for both discs is shown in Fig. 3.4. Note that for the discs, the thrust values are the recorded value from the load cell minus the axial loading due to the rod

\[ T_{\text{disc}} = T_{\text{load cell}} - T_{\text{rod}}, \]  

(3.7)

where \( T_{\text{rod}} \) is calculated using Eq. (3.6). Based on the quadratic fits shown, the thrust coefficients were determined to be \( C_t = 1.08 \) and \( C_t = 0.91 \) for the 45% porous and the 50% porous discs, respectively.

3.6 Wake profile

Subsequently, the ADV was used to record the wake profile behind the 45% porous disc inside the flume with the same flow conditions as those described above, section 3.4. Contour plots for the laboratory experiments corresponding to measurements recorded on a plane at hub height are shown in Fig. 3.5. Measurements start at 3\( D \) downstream of the disc and extend to 11.5\( D \).

The lateral expansion of the wake is limited to within \( \pm D \) and the flow outside this range is not significantly affected by the presence of the disc. However, in the streamwise direction, even at the furthest measured point away from the disc along the centreline, the velocity in the wake has only reached 80% of the freestream value. In the turbulence intensity profile, the two bands of high \( I \) observed are caused by the shear layer which forms at the edges of the disc (\( \pm D/2 \)) and extends downstream until it diffuses away as the wake starts to re-energise. Furthermore, both velocity deficit and turbulence intensity profiles appear symmetric about the centreline, as expected.

In order to elaborate on how this wake profile was obtained, two example ADV time series are shown in Fig. 3.6. The raw data are shown in blue and the post-processed data are shown in red. Fig. 3.6a corresponds to a measurement taken in the bypass flow region away from the disc and Fig. 3.6b corresponds to a measurement taken within the shear layer that develops downstream of the disc between the wake and the bypass flow. In both cases, the majority of the datapoints rejected exhibit
Chapter 3: ADM-RANS validation: Flow past a porous disc

Figure 3.5: Contour plots showing the flow past the 45% porous disc ($C_t = 1.08$) recorded on a plane at hub height behind the disc. × marks the locations where the ADV was used to record measurements.

$R^2$ values below the 70% threshold. This is illustrated in Fig. 3.7. Some further data points have been rejected since they lie more than three standard deviations away from the mean velocity, which has helped remove any remaining velocity spikes.

Within the shear layer, the high velocity bypass flows mixes with the almost stagnant fluid in the wake. Hence, the relative velocity fluctuations at this location exhibit a greater magnitude than those observed for the time series recorded in the bypass flow region. As previously described in section 3.3.2, these fluctuations are used to compute $I_{exp}$. Hence, a value of $I_{exp} = 6.4\%$ was obtained for the bypass flow measurement and a value of $I_{exp} = 16.6\%$ was computed for the shear layer measurement. This is also reflected in the difference in the spread of the set of dat-
apoints shown in Fig. 3.7. Note that these $I_{\text{exp}}$ are computed based on the velocity fluctuations in all three components, but only the streamwise component is shown in Fig. 3.6–3.7.

![Figure 3.6: ADV time series at two different locations downstream of the porous disc. The raw data is shown in blue and the post-processed data is shown in red.](image)

(a) Bypass flow ($x = 3D$, $y = 2D$): $\overline{u_x} = 0.264 \text{ m s}^{-1}$, $I_{\text{exp}} = 6.4\%$

(b) Shear layer ($x = 3.5D$, $y = 0.5D$): $\overline{u_x} = 0.195 \text{ m s}^{-1}$, $I_{\text{exp}} = 16.6\%$

**Figure 3.6:** ADV time series at two different locations downstream of the porous disc. The raw data is shown in blue and the post-processed data is shown in red.

![Figure 3.7: ADV correlation data at two different locations downstream of the porous disc. The raw data is shown in blue and the post-processed data is shown in red.](image)

(a) Bypass flow ($x = 3D$, $y = 2D$)

(b) Shear layer ($x = 3.5D$, $y = 0.5D$)

**Figure 3.7:** ADV correlation data at two different locations downstream of the porous disc. The raw data is shown in blue and the post-processed data is shown in red.
3.7 ADM-RANS simulation

The Fluidity ADM-RANS model was verified against a simple and well-established reference solution in section 2.3. In this section, a much more realistic test case is considered to assess the suitability of the RANS models employed. Having established the wake profile behind the porous disc experimentally, the laboratory scenario is replicated numerically and the ADM-RANS model is used to compute the wake profile. Fig. 3.8 illustrates the 3D domain used in the numerical simulations.

The $C_t$ value used in the ADM-RANS model is limited to a maximum value of 1. This is due to the computation of upstream velocity, $u_0$, which is based on the local velocity at the actuator disc, Eq.(3.8), which itself depends on Eq. (2.8). Alternatively, Eq.(3.8) can be avoided and $u_0$ set equal to the inlet velocity value, $u_{in}$. However, the correction terms used in the modified $k - \omega$ model, Eq. (2.6) and Eq. (2.9), also depend on Eq. (2.8) and therefore cannot be used together with a $C_t$ value greater than 1. Hence, $C_t = 1$ has been used in the simulations which is 7.4%
smaller than the experimentally measured value of $C_t = 1.08$ (section 3.5). This can be a source of discrepancy and will be discussed later in section 3.7.3.

### 3.7.1 Boundary conditions

For the comparisons against the experimental setup, the side walls and the top surface are all set to free slip and the inlet is a Dirichlet boundary condition with constant inlet values, $u_{in}$, $k_{in}$ and $\omega_{in}$. A zero flux boundary condition has been applied at the walls for both $k$ and $\omega$ and a zero pressure boundary condition has been applied at the outlet. When attempting to simulate the flow past the porous disc inside the flume, it is important to capture the vertical asymmetry caused by the slower moving fluid near the sea bed. Hence, in the velocity field a quadratic drag boundary condition is applied to the bottom surface. In the Fluidity finite element discretisation, integration by parts is applied to obtain the weak form of the RANS equations, Eq. (2.1). This leads to a boundary integral for the stress term. For the quadratic drag boundary condition, the tangential component of this integral is set equal to $C_D |u||u|$, whereas for the free slip boundary condition, this is simply set equal to zero (Applied Modelling and Computation Group (AMCG), 2015). This is similar to the bottom boundary condition used in Roc et al. (2013). The bottom drag coefficient, $C_D$, value commonly used in marine simulations is $C_D = 0.0025$ (Vennell, 2010; Divett et al., 2013; Funke et al., 2014; Martin-Short et al., 2015) and this is the value chosen for this study.

Furthermore, $u_0$ has been computed here using the velocity at the disc, $u_{disc}$, and the axial induction factor, $a$, Eq. (2.8),

$$u_0 = \frac{u_{disc}}{1 - a},$$

where $u_{disc}$ is calculated by computing the average streamwise component of velocity of the elements making up the disc.

It is vital to ensure that the ambient conditions of the experimental flume are maintained in the simulations. The ambient turbulence intensity as well as the velocity have to be similar to that of the flume and the inlet values selected must reflect the experimental data. The values $u_{in} = 0.27 \text{ m s}^{-1}$ and $k_{in} = 4 \times 10^{-4} \text{ m}^2 \text{s}^{-2}$ are chosen.
given that these are the recorded ambient values. In the absence of an appropriate bottom boundary condition, the inlet $I$ would decay downstream and is dependent on the inlet $\omega$ value. In the simulations, $I$ is computed via

$$I = \frac{\sqrt{\frac{2}{3}k}}{u_{in}},$$

(3.9)

![Graph showing variation of $I$ in the streamwise direction](image)

**Figure 3.9:** The effect of varying $\omega_{in}$ on the velocity and turbulence intensity values in the 4ft flume simulations in the absence of any turbines.

Fig 3.9a displays the variation of $I$ in the streamwise direction for different $\omega_{in}$ values. Evidently, the quadratic drag boundary condition leads to minimal bottom turbulence generation and is not sufficient to help maintain $I$ in the streamwise direction. A better representation can be achieved via the use of wall functions for the
3.7: ADM-RANS simulation

$k$ and $\omega$ fields similar to those available in commercial CFD packages, e.g. Ansys CFX, and used in Harrison et al. (2010) and Shives and Crawford (2014). These are currently unavailable in the Fluidity model, but are being actively pursued.

Fig. 3.9 also displays velocity and turbulence intensity depth profiles for the empty flume simulations alongside the experimental data. There is very little difference in the velocity profiles outside the boundary layer (Fig. 3.9b). Furthermore, the $I$ profiles have not deviated significantly from the experimental data and all lie within the 5–6% range (Fig. 3.9c). It has been shown in Fig. 3.9a that high $\omega_{in}$ will result in a significant drop in $I$ in the streamwise direction. Therefore $\omega_{in} = 0.25 \text{s}^{-1}$ has been chosen to ensure that both velocity and TKE do not deviate significantly from the values specified at the inlet.

3.7.2 ADM-RANS vs experiments

Fig. 3.10 displays the numerical results using the ADM-RANS model with the modified $k-\omega$ model (section 2.2.3) and the $k-\omega$ SST model (section 2.2.4) alongside the experimental flume results obtained using the 45% porous disc. The closest match in velocity deficit is achieved by using the $k-\omega$ SST model where the simulated results closely follow the experimental data. However, this model slightly underestimates the peak $I$ value. Furthermore, the rate of decay of turbulence intensity in the streamwise direction is greater than that observed in both the experimental data and the $k-\omega$ model results, Fig. 3.10b.

A sudden drop in $I$ values is observed downstream of the disc in the modified $k-\omega$ model profiles due to the correction terms introduced in section 2.2.3. These are introduced to help capture the short-circuiting of the turbulence cascade due to the disc. Without these terms the magnitude of $I$ would be overpredicted resulting in a quicker wake recovery. The results illustrate that increasing $C_\omega$, Eq. (2.9), allows the $k-\omega$ model to better predict the experimental velocity deficit by increasing the peak deficit and shifting it further downstream, Fig. 3.10a. The reason for this lies in the fact that a higher $C_\omega$ results in increased $\omega$ which dissipates the TKE leading to a drop in the peak $I$ values, Fig. 3.10b, and this delays the wake recovery. Although $C_\omega = 8$ better predicts the velocity deficit and captures the correct TKE patterns, the $I$ values move away from the experimental ones as $C_\omega$ is increased.
Chapter 3: ADM-RANS validation: Flow past a porous disc

Figure 3.10: Comparison plots along the centreline \((y = 0, z = 0)\) for the various Fluidity ADM-RANS simulations vs the experimental data.

Figure 3.11: Lateral comparison plots for the various Fluidity ADM-RANS simulations vs the experimental data at \(x = 5D\) and \(x = 10D\). The same legend as in Fig. 3.10 has been used.
The velocity deficit and turbulence intensity comparisons in the lateral direction at \( x = 5D \) and \( x = 10D \) are also presented in Fig. 3.11. The double peak is due to the TKE ring that forms around the disc in the region separating the wake from the bypass flow where mixing occurs, Fig. 3.12. This was also visible in the experimental wake profile, Fig. 3.5, and has been captured by the ADM-RANS model. However, in all of the simulations, the numerical model has slightly underestimated the wake widths. While increasing \( C_\omega \) in the \( k-\omega \) model leads to greater peak velocity deficit values, it has negligible effect on the wake widths. This suggests that the ADM-RANS model underestimates the amount of mixing in the lateral direction and the additional source terms included only help encourage mixing in the streamwise direction.

**Figure 3.12:** The TKE downstream of the disc is shown at \( 2D, 6D \) and \( 10D \) downstream of the disc. A ring of TKE forms behind the actuator disc which dissipates away downstream as the wake recovers.

The root mean square error (RMSE) between the simulated lateral profiles at \( x = 3D, 4D, ..., 11D \) and the experimental wake profile, Fig. 3.5, have also been computed and are presented in Fig. 3.13. Once again, the \( k-\omega \) SST model provides the closest match to the velocity deficit values and captures the correct turbulence intensity profiles, but underestimates the magnitude of \( I \). On the other hand, the modified \( k-\omega \) model with \( C_\omega = 2 \) provides a closer match to the turbulence intensity data, but underestimates the velocity deficit values, both in the near and far wake.
Chapter 3: ADM-RANS validation: Flow past a porous disc

3.7.3 $C_t$ sensitivity

As mentioned at the beginning of this section, the $C_t$ value used in the ADM-RANS simulations is smaller than the value obtained experimentally. To explore how this will affect the results, a brief $C_t$ sensitivity study is presented here. The Fluidity ADM-RANS $k - \omega$ SST model has been considered with three different $C_t$ values of 0.9, 0.95 and 1 and the boundary conditions have remained the same as before. Fig. 3.14 illustrates the effect of varying $C_t$ on the velocity deficit and the turbulence intensity observed in the streamwise direction along the centreline.

![Figure 3.13: RMSE values in velocity deficit and turbulence intensity for the lateral profiles at $x = 2D, 3D, ... , 12D$ comparing the various ADM-RANS simulations to the experimental data. The same legend as in Fig. 3.10 has been used.](image)

![Figure 3.14: Comparison plots along the centreline ($y = 0, z = 0$) for the Fluidity ADM-RANS simulations vs the experimental data with different $C_t$ values.](image)

Evidently, varying the $C_t$ has a significant effect on the velocity deficit in the near wake region where a 10% drop in $C_t$ leads to $\sim 30\%$ drop in peak velocity deficit. However, the profiles are comparable at around $5D$ and these differences are neg-
ligible in the far wake profiles where all three simulations closely follow the experimental data. This trend is also visible in the turbulence intensity profiles where the differences are greatest in the near wake, but are drastically reduced producing similar profiles beyond $10D$. Hence, it appears that by using a $C_t$ value which is 7.4% smaller than the experimental value, the velocity deficit and turbulence intensity values in the near wake are underestimated, but the far wake profiles are not significantly affected.

### 3.7.4 Disc thickness sensitivity

The ADM theory outlined by Houlsby is two-dimensional and therefore assumes a disc with zero thickness. This is clearly not feasible in real world applications, but in order to minimise the effects of disc thickness, in the experiments, the disc had a thickness of 0.6 mm ($0.004D$). In the 3D ADM-RANS model the disc has a finite thickness and for practical reasons is larger than in the experiments. It is therefore necessary to determine how the thickness of the disc will affect the results of the numerical simulations.

![Figure 3.15: Comparison plots along the centreline ($y = 0, z = 0$) for the Fluidity ADM-RANS simulations vs the experimental data with different disc thickness values and $C_t = 1$.](image)

The Fluidity ADM-RANS $k - \omega$ SST model is used with three different thickness values of $D/4$, $D/8$ and $D/12$. The boundary conditions have remained the same as before and $C_t = 1$. Examining the velocity deficit and turbulence intensity profiles along the centreline, Fig. 3.15, reveals that varying the thickness has a
negligible effect on the wake profiles. The only observable difference occurs in the peak turbulence intensity values, but even these are only present between $4D$ to $6D$. The far wake profiles are almost identical.

### 3.7.5 Mesh optimisation

All the numerical simulations presented in this chapter have utilised mesh optimisation techniques in order to ensure an accurate solution without the need for excessive computational resources. The accuracy of the mesh optimisation techniques employed have to be assessed to ensure that using this capability has not jeopardised the accuracy of the numerical solutions. Hence, the experimental flume setup is simulated using the $k-\omega$ SST model with a range of fixed meshes of varying resolution and the results are compared against the *adaptive* mesh results. Table 3.3 outlines the properties of the different meshes considered. Note that with increasing mesh resolution, the disc thickness of the fixed meshes is also reduced. However, as demonstrated above, this has a negligible effect on the resulting wake profiles. For the purpose of the mesh optimised simulations, the mesh is refined in regions of high curvature in the velocity, $k$, and $\omega$ fields, and the minimum and maximum values of the element edge length were set to $l_e/D = 1/12$ and $l_e/D = 2$, respectively. A timestep of 0.05 s was used and the mesh was optimised after every 20 timesteps where consistent interpolation was used to transfer the data from the old mesh onto the new optimised mesh, section 2.2.5.

<table>
<thead>
<tr>
<th>Simulation</th>
<th>max $l_e/D$</th>
<th>min $l_e/D$</th>
<th>$t/D$</th>
<th>No. vertices</th>
<th>No. elements</th>
<th>CPU hours</th>
</tr>
</thead>
<tbody>
<tr>
<td>very coarse</td>
<td>1</td>
<td>1/4</td>
<td>1/4</td>
<td>12,103</td>
<td>63,360</td>
<td>26</td>
</tr>
<tr>
<td>coarse</td>
<td>1/2</td>
<td>1/8</td>
<td>1/8</td>
<td>53,934</td>
<td>288,876</td>
<td>117</td>
</tr>
<tr>
<td>fine</td>
<td>1/3</td>
<td>1/12</td>
<td>1/12</td>
<td>184,460</td>
<td>1,029,600</td>
<td>471</td>
</tr>
<tr>
<td>very fine</td>
<td>1/4</td>
<td>1/16</td>
<td>1/16</td>
<td>397,618</td>
<td>2,250,360</td>
<td>1284</td>
</tr>
<tr>
<td>adaptive</td>
<td>2</td>
<td>1/12</td>
<td>1/12</td>
<td>98,537*</td>
<td>547,198*</td>
<td>378</td>
</tr>
</tbody>
</table>

Table 3.3: Properties of the different meshes used. *These are the maximum values use in the *adaptive* simulation.

The main aim in this section is to establish that the *adaptive* mesh results have not deviated from the *mesh converged* solution. Fig. 3.16 displays the velocity deficit
and turbulence intensity profiles along the centreline for all the fixed mesh simulations, alongside the adaptive mesh results. The experimental data are also shown, although it is not of a primary concern in this section, it helps put the magnitude of the variations between the different solutions into context.

In the velocity deficit profiles, the greatest deviation occurs with the very coarse mesh. Here, the velocity deficit values are exaggerated in the near wake, but in the far wake the predicted values are smaller than those predicted by the higher resolution meshes. This suggests that the low resolution results in greater spurious diffusion which has led to a faster wake recovery. The turbulence intensity profiles reveal that the very coarse model overpredicts the $I$ values with a grossly exaggerated single peak, as opposed to the slightly delayed peak observed with all the other solutions. The delayed peak profile occurs due to the short-circuiting of the turbulence cascade at the disc which in the $k-\omega$ SST is modelled via the $F_1$ blending function, discussed previously in section 2.2.4. The low resolution model has completely missed this, leading to much higher $I$ values in the wake.

![Figure 3.16: Mesh sensitivity plots along the centreline ($y = 0, z = 0$).](image)

All the other meshes produce very similar results both for velocity deficit and turbulence intensity. However, in order to better quantify the differences, the very fine mesh solution is judged to be the mesh converged solution and the RMSE values between the various fixed and adaptive mesh solutions relative to this converged solution have been computed, Table 3.4. The adaptive mesh solution provides the closest match in velocity deficit, but it does not perform as well in modelling the
Chapter 3: ADM-RANS validation: Flow past a porous disc

... turbulence intensity values along the centreline. This deviation is due to the greater peak $I$ values observed around $4D$, Fig. 3.16b.

<table>
<thead>
<tr>
<th>Simulation</th>
<th>very coarse</th>
<th>coarse</th>
<th>fine</th>
<th>adaptive</th>
</tr>
</thead>
<tbody>
<tr>
<td>$1 - u_x/u_0$</td>
<td>0.0995</td>
<td>0.0223</td>
<td>0.0119</td>
<td>0.0101</td>
</tr>
<tr>
<td>$I$</td>
<td>0.0264</td>
<td>0.0026</td>
<td>0.0018</td>
<td>0.0035</td>
</tr>
</tbody>
</table>

Table 3.4: RMSE values in velocity deficit and turbulence intensity comparing the various simulations to the mesh converged (i.e. very fine mesh) solution. The corresponding wake profiles are shown in Fig. 3.16.

Figure 3.16: Mesh sensitivity velocity deficit and $I$ plots in the lateral direction at $x = 5D$ and $x = 10D$. The same legend as in Fig. 3.16 has been used.

The RMSE values shown in Table 3.4 have only focused on a single line within the 3D domain. For a more comprehensive analysis, the effect of mesh resolution on the various lateral profiles at hub height are also examined. Fig. 3.17 displays lateral profiles of velocity deficit and turbulence intensity at $x = 5D$ and $x = 10D$. The
3.7: ADM-RANS simulation

differences are much more visible now, with the *very coarse* resolution underestimating the velocity deficit, despite the exaggerated values observed in the near wake, Fig. 3.16a. At $x = 5D$, the *adaptive* mesh solution slightly underpredicts and the *coarse* and *fine* mesh solutions slightly overpredict the velocity deficit values relative to the *mesh converged* (i.e. *very fine* mesh) solution. These differences are not visible at $x = 10D$ where all meshes, apart from the *very coarse* mesh, produce very similar profiles.

In the turbulence intensity profiles, the exaggerated diffusion in the low resolution computation leads to a misrepresentation of the double peak profile. Interestingly, this double peak becomes more refined with increasing mesh resolution. Once again, in order to look at the differences in greater detail, RMSE values between the various fixed and *adaptive* mesh solutions and the *mesh converged* (i.e. *very fine* mesh) solution have been computed. The lateral profiles at $x = 2D, 3D, ..., 12D$ have been considered and the results are presented in Fig. 3.18. These plots demonstrate that the *adaptive* mesh solution closely follows the *fine* mesh solution and that both of these solutions closely mimic the *mesh converged* solution.

![Figure 3.18: RMSE values in velocity deficit and turbulence intensity for the lateral profiles at $x = 2D, 3D, ..., 12D$ comparing the different meshes against the *mesh converged* (i.e. *very fine* mesh) solution. The same legend as in Fig. 3.16 has been used.](image)

Moreover, despite the computational time taken by the adaptivity algorithm, the *adaptive* mesh solution required 378 CPU hours compared with 471 CPU hours and 1284 CPU hours required by the *fine* mesh and *very fine* mesh solutions, respectively. Please note that this does not include the pre-processing time spent on creating suitable fixed meshes which was only possible in this case since the location of the wake could be anticipated in advance. Also, note if a uniform fixed mesh had been used with an element edge length equal to the minimum value used in
the adaptive mesh simulation (i.e. $D/12$), then the mesh would have consisted of $6.8 \times 10^6$ elements. This is more than three times the number of elements used in the very fine fixed mesh and the computational cost would have been significantly higher.

### 3.7.6 Reversing flow

In order to demonstrate the benefits of the mesh optimisation capabilities of the Fluidity ADM-RANS model, the flow direction is reversed, once steady state is reached, and the mesh consequently adapts to capture the wake which develops on the other side of the disc. Once again, a timestep of 0.05 s was used and the mesh was optimised after every 20 timesteps where consistent interpolation was used to transfer the data from the old mesh onto the new optimised mesh, section 2.2.5. This is illustrated via 2D slices across the 3D domain at different times throughout the simulation in Fig. 3.19. This can be extended to model transient flows and also flows where the flow direction will not be perfectly aligned with the disc or domain or when localised transient features such as eddies might impinge on the array.
3.7: ADM-RANS simulation

(a) steady state is reached for the flow running from left to right

(b) flow direction is reversed and the mesh optimisation is rearranging the elements in order to capture the wake

(c) steady state is reached for the flow running from right to left

Figure 3.19: Fluidity ADM-RANS model’s mesh optimisation provides an ideal tool to model reversing flows. 2D slices across the 3D domain are shown at several times throughout the simulation.
3.8 Conclusions

At the beginning of this chapter, the necessity to better understand the flow regime downstream of a porous disc representing an energy extracting turbine was outlined. The experiments carried out have been conducted with this aim in mind and in this section the lessons learnt are summarised.

It has been shown that ADM theory presents a robust tool to model the flow downstream of porous discs. The numerical wakes of the Fluidity ADM-RANS model reproduced the main features in the wake that were captured experimentally using the ADV. The shear layer between the wake and the bypass flow where the higher momentum fluid mixes in with the almost stagnant fluid is clearly captured via the ADV, Fig 3.5. The numerical model is also able to capture this and produces a similar wake profile. This is crucial since this mixing is the main mechanism with which the wake is re-energised. The limitations regarding using a porous disc instead of a rotating turbine were discussed in section 3.3.1, but as far as modelling the flow past porous discs is concerned, the ADM-RANS model can accurately predict the flow behaviour at locations far downstream, beyond approximately 6\(D\). The simulations provided a very good agreement in the far wake and given that this study focuses on modelling arrays of turbines, the far wake profiles are of primary importance here.

Furthermore, although ADM theory assumes a disc of zero thickness, it has been shown that the resulting wake profiles are not sensitive to disc thickness and values up to \(D/4\) can be used. This is a significant advantage. If a fine thickness was necessary to correctly capture the wake profile, the resulting mesh requirements would have been much finer and this would have increased the computational cost of the numerical simulations. This is undesirable since the ADM-RANS model has been developed to model multiple rows of tidal turbines and extra care has been taken to ensure the computational costs are kept to a minimum. One of these measures was the dynamic mesh optimisation employed in the Fluidity ADM-RANS model and it has been demonstrated that using this feature can help improve the accuracy of the numerical simulations whilst minimising the computational costs.
ADM-RANS validation: 
Flow past multiple scaled turbines

The test cases presented in this chapter are derived from and expand upon a journal paper published in the Journal of Fluids and Structures by Abolghasemi et al. (2016b). The material therein has arisen as the result of the first author’s research and the co-authors have acted in guiding roles only.

4.1 Introduction

The performance of the ADM-RANS model presented in this thesis was analysed against a well-established reference semi-analytical solution in section 2.3. Moreover, the numerical model was used to recreate a laboratory setup of a flow past a single porous disc rotor simulator in section 3.7. Therein, the performance of the model against experimental data was thoroughly examined and the limitations of the model were identified. The suitability of the RANS models implemented and the mesh optimisation techniques used were also explored.

In this chapter, the Fluidity ADM-RANS model is used to recreate two more realistic
test cases involving interacting multiple scaled turbines. In the previous chapter the ADM-RANS model performed well when having to simulate the flow past a porous disc. However, the two test cases considered here use scaled rotors. Hence, there will be a swirl element in the wake downstream of the turbines which the ADM-RANS model neglects. In the formulation of the ADM-RANS model it was assumed that the swirl only affects the near wake profile and that the far wake is essentially unaffected. The test cases considered here will test that hypothesis. Furthermore, the ability of the model to correctly simulate wake interactions is also assessed as this will be critical in array scale simulations of multiple turbines.

4.2 Stallard et al. test case

The first test case considered is an experimental study of a group of three-bladed rotors carried out by Stallard et al. (2013). These experiments were conducted in the University of Manchester wide flume, which is 5 m wide with a 12 m long test section. The tests utilised fixed pitch rotors with a diameter $D = 0.27$ m which were located at mid-depth and positioned 6 m away from the inlet. The Fluidity ADM-RANS model has been used to simulate the flow past the three scaled turbines positioned in-line, with the numerical domain used shown in Fig. 4.1. Previously, Mungar (2014) used a turbine model consisting of a momentum sink term developed within the Delft3D framework (Delft Hydraulics, 1999), Shives and Crawford (2016) used a BEM model implemented in Ansys CFX (Ansys Inc., 2015) and Olczak et al. (2016) implemented a RANS-BEM model in the commercial CFD code StarCCM+ to simulate these experiments and their results are briefly discussed later.

4.2.1 Boundary conditions

The boundary conditions used here are similar to those described previously in section 3.7.1 when simulating the flow past a single porous disc. In summary, a Dirichlet boundary condition with constant inlet values, $u_{in}$, $k_{in}$ and $\omega_{in}$ is applied at the inlet and the side walls and top surface are set to free slip. A zero flux boundary condition has been applied at the walls for both $k$ and $\omega$ and a zero pressure boundary condition has been applied at the outlet. In the velocity field a quadratic drag boundary condition is applied to the bottom surface where $C_D = 0.0025$. 

94
4.2: Stallard et al. test case

Figure 4.1: Numerical domain with $D = 0.27 \text{ m}$ used in the Fluidity ADM-RANS simulations of the Stallard et al. experimental setup with $x$, $y$ and $z$ pointing in the streamwise, lateral and vertical directions respectively. The dark grey area represents the actuator discs and the light grey area around the discs represents the locations where the additional $\omega$ source term, Eq. (2.9), is applied.

Furthermore, as in section 3.7.1, since upstream quantities are not always readily available, $u_0$ has been computed here using the velocity at the disc, Eq. (3.8). Note that due to the assumptions made in the derivation of the axial induction factor, $a$, used in Eq. (3.8), this computation of $u_0$ is applicable for low blockage cases only. The scenarios considered here have global blockage ratios, Eq. (1.3), of 2.5% (single turbine) and 7.6% (three turbines), and therefore satisfy this assumption. Having said that, the local blockage ratios, Eq. (1.1), are much higher, especially for the centre disc where $B_L = 31.4\%$ ($B_L = 23.5\%$ for the off-centre discs). This may affect the $C_t$ (Nishino and Willden, 2012a) which will influence the wake interactions downstream of the turbines (Stallard et al., 2013)

4.2.2 Ambient conditions

The inlet values must be chosen carefully in order to ensure the ambient velocity and turbulence properties of the experimental flume are accurately represented via the numerical model. During the experiment, an inflow velocity of $0.47 \text{ m s}^{-1}$ was used
and $I$ at the inlet was reported to be around 11% (Stallard et al., 2013). In the simulations, the values at the inlet were set to $u_{in} = 0.47 \text{ m s}^{-1}$ and $k_{in} = 4 \times 10^{-3} \text{ m}^2 \text{s}^{-2}$. This corresponds to $I_{in} = 11\%$, where $I$ has been computed via Eq. (3.9).

Fig 4.2 displays the variation of $I$ in the streamwise direction for different $\omega_{in}$ values. Once again, it is evident that the quadratic drag boundary condition leads to minimal bottom turbulence generation and even doubling $C_D$ does not do enough to help maintain $I$ in the streamwise direction. As previously discussed in section 3.7.1, a better representation can be achieved via the use of wall functions for the $k$ and $\omega$ fields.

Hence, for this study, in order to ensure that the rate of decay of $k$ is similar to that observed in the experiments, $\omega_{in} = 1.0 \text{ s}^{-1}$ has been chosen. In choosing the $\omega_{in}$ value, one should also bear in mind the implications that this will have on the turbulence length scale, $l$. This can be approximated for both the $k - \omega$ and the $k - \omega$ SST RANS models via

$$ l = \frac{\sqrt{k}}{\beta^* \omega}. $$

Therefore, $\omega_{in} = 0.5 \text{ s}^{-1}$ has been avoided as it suggests $l = 1.41 \text{ m}$ which is more than three times the depth of the flume.

![Figure 4.2: Variation of the turbulence intensity, $I$, along the centreline ($y = 0, z = 0$) for different $\omega_{in}$ values.](image-url)
4.2: Stallard et al. test case

**Figure 4.3:** Ambient depth profile: Fluidity ADM-RANS vs the experimental data where \( x/D = 0 \) is the turbine location.

**Figure 4.4:** Ambient lateral profile: Fluidity ADM-RANS vs the experimental data where \( x/D = 0 \) is the turbine location. The same legend as in Fig. 4.3 is used.

Fig. 4.3–4.4 display ambient velocity deficit and turbulence intensity depth and lateral profiles at two locations along the flume in the absence of turbines. The experimental data is compared against numerical results from the Fluidity ADM-RANS model using both the \( k - \omega \) and the \( k - \omega \) SST RANS models ran on fixed meshes. The numerical data matches the velocity profiles well, but slightly underestimates the turbulence intensity. Both RANS models produce very similar results for the empty flume simulations. This is not surprising because in the absence of a no-slip wall boundary condition, which is the case here, and given that there is no pressure discontinuity in the flow, due to the absence of any turbines, the \( F_1 \) blending function in the \( k - \omega \) SST model equals unity almost everywhere in the domain. Consequently, the \( k - \omega \) SST model behaves very similar to the original \( k - \omega \) model.
4.2.3 Single turbine

Having established a match with the ambient data, initially the flow past a single turbine is considered. In the Fluidity ADM-RANS model $C_t = 0.92$ which corresponds to the upper bound of the $C_t$ value given in Stallard et al. (2013). Stallard et al. (2015) investigated the mean wake properties behind a single three-bladed rotor identical to those considered in Stallard et al. (2013) and demonstrated that for distances greater than $8D$ downstream the velocity deficit becomes two-dimensional and self-similar. The self-similar form follows a Gaussian profile

$$\frac{\Delta u}{\Delta u_{\text{max}}} = \exp\left(-\ln(2) \frac{y^2}{y_{1/2}^2}\right), \quad (4.2)$$

where $\Delta u = u_0 - u_x$ and

$$\frac{\Delta u_{\text{max}}}{u_0} = 0.864(x/D)^{-1/2} - 0.126,$$

$$\frac{y_{1/2}}{R} = 0.412(x/D)^{1/2} + 0.500,$$

where $R = D/2$ is the turbine radius (Stallard et al., 2015). Recently, Stansby and Stallard (2016) used this to demonstrate that the superposition of self-similar velocity profiles with bypass flow selected to ensure mass-flux continuity can lead to reasonably accurate predictions of depth-averaged wake velocities downstream of arrays.

The results from the fixed mesh ADM-RANS $k-\omega$ model, with the aforementioned correction terms, Eq. (2.6) and Eq. (2.9), and the ADM-RANS $k-\omega$ SST model have been compared against this self-similar profile, Fig. 4.5. The ADM-RANS wake profiles also resemble a self-similar solution with a good agreement observed between the lateral profiles at $8D$ and $12D$. There are however discrepancies in the bypass region where the Fluidity ADM-RANS results suggests that the flow is accelerating whereas Eq. (4.2) sets the bypass flow equal to the ambient. Furthermore, both ADM-RANS models predict a narrower profile compared to the experimental data and the self-similar profile. This could be due to the different turbulence length scales in the onset flow in the flume with vertical, lateral and streamwise scales.
4.2: Stallard et al. test case

typically in the ratio 1:3:5 (Stansby and Stallard, 2016). This variation in length scales would not be captured via the RANS models implemented and therefore the model has underpredicted the amount of mixing in the lateral direction. Furthermore, constants in Eq.(4.2) relative to bypass flow velocity assumed equivalent to ambient flow, consistent with accuracy of ADV and measurement range.

4.2.4 Three turbines

The flow past three turbines has also been modelled in Fluidity with mesh optimisation and using the inlet values stated above. In this section the results for the ADM-RANS $k - \omega$ model, with and without the correction terms, Eq. (2.6) and Eq. (2.9), as well as the results for the ADM-RANS $k - \omega$ SST model are presented. Fig. 4.6 displays the variation of velocity deficit, turbulence intensity and non-dimensional length scale, $l/D$, in the streamwise direction along the centreline for the Stallard experimental data and the various ADM-RANS simulations. Note that $l$ is not available for the experimental data since $\omega$ was not experimentally measured.

It is evident that the original $k - \omega$ model underpredicts the velocity deficit significantly. The RANS correction terms, Eq. (2.6) and Eq. (2.9), help improve the
velocity deficit prediction by increasing the peak deficit, Fig. 4.6a, and shifting it further downstream. Previously, in section 3.7.2, it was shown that the peak velocity deficit value can be increased by increasing $C_\omega$, but at the same time this leads to a drop in the peak $I$ value.

**Figure 4.6:** Comparison plots along the centreline ($y = 0, z = 0$) for the Fluidity ADM-RANS simulations vs experimental data from Stallard et al. The results using the $k-\omega$ model with corrections terms is shown in solid blue lines and the results using the original $k-\omega$ model are shown in dotted blue lines. The $k-\omega$ SST results are shown in red. The experimental results are shown in black.
Examining the variation of turbulence length scale, Fig. 4.6c, reveals how the correction terms in the Fluidity ADM-RANS $k-\omega$ model help capture the short circuiting of the turbulence cascade. It also demonstrates how the Fluidity ADM-RANS $k-\omega$ SST model is also able to capture the short circuiting via the blending functions and without the need for additional correction terms. This reflects the popularity of the $k-\omega$ SST model in the literature (Harrison et al., 2010; Afgan et al., 2013; Shives and Crawford, 2015, 2016; Olczak et al., 2016), but to the best knowledge of the author, it is the first time that a comparison between the $k-\omega$ and $k-\omega$ SST model has been presented to demonstrate the advantages of using the latter to simulate the flow past tidal turbine arrays.

Figure 4.7: Lateral velocity deficit, $1-u_x/u_0$, profiles for the ADM-RANS simulations vs Stallard et al. at $x = 2D, 4D, 6D, 8D, 10D$ and $12D$ with $z = 0$. The same legend as in Fig. 4.6 is used.

Figure 4.8: Lateral turbulence intensity, $I$, profiles for the ADM-RANS simulations vs Stallard et al. at $x = 2D, 4D, 6D, 8D, 10D$ and $12D$ with $z = 0$. The same legend as in Fig. 4.6 is used.

The velocity deficit and turbulence intensity comparisons in the lateral direction...
Chapter 4: ADM-RANS validation: Flow past multiple scaled turbines

at various distances downstream of the turbines are also presented in Fig. 4.7–4.8. The RMSE between the simulated results and the experimental data have also been computed and are presented in Fig. 4.9. Although the numerical results predict lower velocity deficit and turbulence intensity values than the experimental data, this match improves further downstream where both the $k - \omega$ ADM-RANS model with correction terms and the $k - \omega$ SST ADM-RANS model show good agreement with the experiments. This is reflected in the reduction of RMSE in the far wake (i.e. $x > 8D$). In fact, even the near wake results are better than those presented in Mungar (2014), which is not surprising since the latter is a depth-averaged model which does not take into account 3D effects. Furthermore, the wake widths have increased compared to those presented in Fig. 4.5 and therefore follow the experimental results closer. It appears that the presence of the additional turbines has encouraged more lateral mixing which has resulted in wider wake profiles.

![Figure 4.9: RMSE values in velocity deficit and turbulence intensity for Fig. 4.7–4.8. The same legend as in Fig. 4.6 is used.](image)

4.2.5 Mesh optimisation

In order to demonstrate the potential advantages of the mesh optimisation available in the Fluidity ADM-RANS model, the adaptive runs are compared against fixed mesh solutions. The fixed mesh adopted for the present test case is illustrated in Fig. 4.10. For the purpose of the adaptive simulations, this mesh is refined in regions of high curvature in the velocity, $k$, and $\omega$ fields, and the minimum and maximum values of the element edge length were set to $l_e/D = 0.037$ and $l_e/D = 1.11$ respectively. The minimum edge length specified is smaller than the finest resolution of $l_e/D = 0.055$ used in the fixed mesh and the maximum edge length used in the
adaptive simulation is slightly larger than the rotor diameter and three times greater than the coarsest resolution of $l_e/D = 0.370$ used in the fixed mesh. Both fixed and adaptive mesh simulations ran until steady state was achieved and the resulting wake profiles were compared to ensure no loss of accuracy as a result of using mesh optimisation, Fig. 4.11. The RMSE between the wake profiles of Fig. 4.11 and the experimental results are presented in Table 4.1, illustrating a close agreement between the wake profiles of the fixed and adaptive mesh simulations.

**Figure 4.10:** A cut through the mesh used for the fixed mesh simulation with the discs positioned in the centre. A finer resolution of $l_e/D = 0.055$ is used around the discs with a coarse resolution of $l_e/D = 0.370$ used in the periphery. This mesh has been regularly extruded in the streamwise direction with a layer thickness of $D/8$ (0.03375 m) which is equal to the disc thickness.

<table>
<thead>
<tr>
<th>Mesh type</th>
<th>$1 - u_x/u_0 (2D, 6D, 12D)$</th>
<th>$I (2D, 6D, 12D)$</th>
</tr>
</thead>
<tbody>
<tr>
<td>fixed</td>
<td>0.128, 0.064, 0.053</td>
<td>0.056, 0.019, 0.0075</td>
</tr>
<tr>
<td>adaptive</td>
<td>0.130, 0.065, 0.052</td>
<td>0.054, 0.017, 0.0072</td>
</tr>
</tbody>
</table>

**Table 4.1:** RMSE values in velocity deficit and turbulence intensity comparing the fixed and adaptive mesh simulation results to the experimental data. The corresponding wake profiles are shown in Fig. 4.11.

The fixed mesh simulation used $3.8 \times 10^5$ vertices compared to a maximum of only $1.0 \times 10^5$ vertices used in the adaptive simulation. Despite the computational time taken up by the adaptivity algorithms, section 2.2.5, the adaptive mesh simulation still takes $\sim 32$ hours compared to $\sim 71$ hours needed for the fixed mesh simulation when both simulations were run in parallel on 16 cores. This adaptive runtime corresponds to $\sim 512$ CPU hours which is an order of magnitude greater than the $\sim 30$ CPU hours reported for the fixed mesh BEM model of Malki et al. (2013) to simulate the flow past a single turbine. However, the Malki et al. (2013) model is a steady state finite volume model with a carefully constructed mesh consisting of $1.5 \times 10^6$ elements which uses a fine resolution close to the turbine. Hence, extending the model to simulate the flow past tidal turbine arrays requires the construction...
of an adequate mesh prior to the simulations. On the other hand, the dynamic mesh optimisation capabilities of the Fluidity model can automatically adjust the mesh to ensure the physics is accurately captured, regardless of the number of turbines, which saves time on the pre-processing as well as the overall runtime. Furthermore, the Fluidity runtime is only a fraction of the 0.14 million CPU hours reported for the moving mesh RANS model of Afgan et al. (2013), although the latter is an unsteady RANS model that resolved individual blades and the substantial runtime is predominantly due to the constraint on timestep where a value of $2 \times 10^{-5}$ s had to be used (Afgan et al., 2013).

![Image](image_url)

**Figure 4.11:** Lateral velocity deficit and turbulence intensity plots at $x = 2D$, $x = 6D$ and $x = 12D$ with $z = 0$ showing the results from both the fixed mesh and the adaptive mesh simulations using the ADM-RANS $k - \omega$ SST model with $C_t = 0.92$.

An important point to note is that in this simple case, since the location of the wake can be predicted, a non-uniform mesh has been carefully created for the fixed mesh ADM-RANS simulation which only uses a fine resolution in locations of interest. However, the location and extent of the wake is not always known a priori in the real world, e.g. due to time dependent variations in the tidal flow, and a high resolution uniform mesh would be required potentially everywhere. In this extreme case, if the finest resolution was used throughout the domain, this would have led to $2.8 \times 10^6$ vertices. This is more than seven times larger and would have required
significantly more computational power to achieve the same level of accuracy as the adaptive mesh simulation. This would of course be further exacerbated if a larger domain and a large array of turbines was considered.
Figure 4.12: Results from an ADM-RANS $k-\omega$ SST simulation of the Stallard et al. experimental setup is shown. During the ADM-RANS simulation dynamic mesh optimisation has been used to adapt to the velocity, $k$ and $\omega$ fields. A 2D slice across the 3D domain is presented here showing the velocity and $k$ fields at hub height ($z = 0$) along with the adapted mesh.
4.3 Mycek et al. test case

In order to further validate the ADM-RANS model presented, a comparison between the numerical model and results from an experimental study of a pair of three-bladed 1/30th scale prototypes of horizontal axis tidal turbines has also been conducted. In this case, the turbines are axially aligned one behind another as opposed to the single-row arrangement of the previous test case. The experimental dataset chosen is taken from a study by Mycek et al. (2014a,b) where the effect of turbulence intensity on the wake structure of the turbines was examined. Mycek et al. (2014a,b) conducted experiments at two different background turbulence intensities, \( I_\infty = 3\% \) and \( I_\infty = 15\% \), with similar inflow conditions. The two turbines used in both cases were running at a fixed tip speed ratio, \( \text{TSR} = 3.67 \), relative to the onset flow of the front turbine (Mycek et al., 2014b). Using the thrust curves provided, the corresponding thrust coefficients were determined as \( C_t = 0.81 \) \( (I_\infty = 3\%) \) and \( C_t = 0.75 \) \( (I_\infty = 15\%) \) (Mycek et al., 2014a).

![Figure 4.13: Numerical domain with \( D = 0.792 \text{m} \) used in the Fluidity ADM-RANS simulations of the Mycek et al. experimental setup with \( x, y \) and \( z \) pointing in the streamwise, lateral and vertical directions respectively. The dark grey area represents the actuator discs and the light grey area around the discs represents the locations where the additional \( \omega \) source term, Eq. (2.9), is applied.](image)

Initially the wake behind the upstream turbine, in the absence of the downstream turbine, was recorded and then the wake behind the second turbine, positioned 6\( D \).
downstream of the first turbine, was also recorded. This approach was emulated in the numerical study and Fig. 4.13 shows the numerical domain used in Fluidity ADM-RANS which is based on the dimensions of the IFREMER flume where the experiments were conducted. The same boundary conditions previously described in section 4.2.1 have also been applied here, and Table 4.2 describes the inlet conditions used in the Fluidity ADM-RANS simulations. The rate of decay of turbulence in the streamwise direction was not available in this case and the $\omega_{in}$ value chosen has been motivated by the values used in Shives and Crawford (2015). The blockage ratio in this case is 6.2% and as with the previous test case, Eq. (3.8) has been used to compute $u_0$.

<table>
<thead>
<tr>
<th>Inlet property</th>
<th>$I_\infty = 3%$</th>
<th>$I_\infty = 15%$</th>
</tr>
</thead>
<tbody>
<tr>
<td>$u_{in}$</td>
<td>0.80 m s$^{-1}$</td>
<td>0.83 m s$^{-1}$</td>
</tr>
<tr>
<td>$k_{in}$</td>
<td>0.001 m$^2$ s$^{-2}$</td>
<td>0.025 m$^2$ s$^{-2}$</td>
</tr>
<tr>
<td>$\omega_{in}$</td>
<td>0.50 s$^{-1}$</td>
<td>0.50 s$^{-1}$</td>
</tr>
<tr>
<td>$I_{in}$</td>
<td>3.23%</td>
<td>15.55%</td>
</tr>
</tbody>
</table>

**Table 4.2:** Fluidity ADM-RANS simulation inlet conditions for the Mycek et al. test cases. The velocity values are taken from Mycek et al. (2014a) and the $k$ values are chosen to give turbulence intensity values similar to those recorded experimentally.

The results from the adaptive mesh simulation are shown in Fig. 4.14–4.15. The Fluidity ADM-RANS model has successfully captured the increased rate of wake recovery expected as a result of the increase in background turbulence intensity. In fact for the $I_\infty = 15\%$ case the wake of the upstream turbine is almost fully recovered $6D$ downstream where the second turbine is positioned and both turbines produce similar wake profiles. This is definitely not the case with $I_\infty = 3\%$ where the rate of wake recovery is much slower and the second turbine is placed in the wake of the upstream turbine which has also resulted in a greater velocity deficit.

Fig. 4.16 shows lateral velocity deficit and turbulence intensity profiles behind the upstream turbine for the various Fluidity ADM-RANS simulations of the low background turbulence intensity case, with the high turbulence intensity case shown in Fig. 4.17. Similar velocity deficit and turbulence intensity profiles behind the
4.3: Mycek et al. test case

(a) streamwise component of velocity

(b) turbulent kinetic energy $k$

(c) a close up of the adapted mesh near the actuator discs (shown in grey)

Figure 4.14: Results from an ADM-RANS $k-\omega$ SST simulation of the Mycek et al. experimental setup is shown. A 2D slice across the 3D domain is presented here showing the velocity and $k$ fields at hub height ($z = 0$) along with the adapted mesh for the $I_\infty = 3\%$ case with $C_t = 0.81$ and $u_{in} = 0.80 \text{ m s}^{-1}$.

downstream turbine are shown in Fig. 4.18–4.19. Note that in the experimental study Mycek et al. presented profiles of turbulence intensity normalised by the local velocity based on only two of the three velocity components,

$$I_{2D} = \sqrt{\frac{1}{2} \left( \frac{u'_x u'_x + u'_y u'_y}{u_x^2 + u_y^2} \right)}$$  \hspace{1cm} (4.3)

where $u'_x$ is the instantaneous turbulent velocity fluctuations in the $x$ direction and similarly $u'_y$ is the instantaneous turbulent velocity fluctuations in the $y$ direction. Shives and Crawford (2015) also conducted a numerical study of these experiments and describes how the experimental $I_{2D}$ values can be used to derive $I$ values normalised by the upstream velocity, Eq. (3.9), which take into account all three velocity components. Given that the TKE in the RANS models is based upon all three components of velocity, the experimental $I$ values presented here are normalised following the same procedure as Shives and Crawford (2015).
Chapter 4: ADM-RANS validation: Flow past multiple scaled turbines

Figure 4.15: Results from an ADM-RANS $k-\omega$ SST simulation of the Mycek et al. experimental setup is shown. A 2D slice across the 3D domain is presented here showing the velocity and $k$ fields at hub height ($z = 0$) along with the adapted mesh for the $I_\infty = 15\%$ case with $C_t = 0.75$ and $u_{in} = 0.83 \text{ m s}^{-1}$.

As with the Stallard test case, mesh optimisation has been used throughout these simulations to refine the mesh in regions of high curvature in the velocity, $k$, and $\omega$ fields and the minimum and maximum values of the element edge length were set to $l_e/D = 0.063$ and $l_e/D = 0.631$ respectively. These $l_e/D$ values are not refined enough for accurate study of turbine blade turbulence. However, given that the actual turbine blades are not being modelled here, the $l_e/D$ values chosen are adequate for the ADM-RANS model adopted. Eddies are not modelled in the RANS models employed, if an LES approach had been followed, then a far finer resolution would certainly have been necessary. Also, note that the maximum edge length is only a factor of ten greater than the minimum which was found to be sufficient for accurate wake characterisation in this case. In order to ensure a smooth transition from small elements to larger elements, mesh gradation algorithms are used in Fluidity that constrain the rate of growth in desired edge lengths along an edge. Throughout this study, a conservative gradation parameter value of 1.5 has been used. The
limiting factor regarding this value and the ratio of the minimum to maximum edge length depend on the numerical schemes used and the physics involved. For larger scale simulations these can be extended to higher values without jeopardising the accuracy of the simulations, although this has not been explored in this study.

As observed previously in section 4.2.4, the original $k - \omega$ model over-predicts the rate of wake recovery for every case considered, Fig. 4.16–4.19. In contrast, with the help of the correction terms this is delayed and the model is able to follow the experimental results much better. Once again, a good agreement with the experimental data is produced via the $k - \omega$ SST model for both low and high turbulence intensity cases without the need to include additional turbulence correction terms. Furthermore, there is very little difference between the output from the $k - \omega$ SST model and the $k - \omega$ model with correction terms, especially in the far wake. These results are consistent with Shives and Crawford (2015) where an ADM model coupled with $k - \omega$ SST RANS was used. The experimental data for the $I_\infty = 15\%$ case

![Graph showing lateral velocity deficit and turbulence intensity profiles](image)

**Figure 4.16:** Lateral velocity deficit and turbulence intensity profiles for the ADM-RANS simulations vs Mycek et al. at $x = 2D$, $4D$, $6D$, $8D$ and $10D$ for the upstream turbine with $I_\infty = 3\%$, $C_t = 0.81$ and $u_\infty = 0.80 \text{ m s}^{-1}$. The same legend as in Fig. 4.6 is used.
appears to be skewed towards one side of the tank and Mycek et al. (2014a) put this down to the removal of the flow conditioning screens from the flume, which was necessary to achieve the more turbulent flow. Given that this was not modelled in the numerical simulations, since sufficient data regarding the inflow was not available, the outputs from the ADM-RANS models do not agree as well as the $I_\infty = 3\%$ case. That being said, a good agreement can be seen from $6D$ onwards in both velocity deficit and turbulence intensity values for the upstream turbine, Fig. 4.17, as well as the downstream turbine, Fig. 4.19. In comparison to the previous test case, a closer agreement between the simulated and experimental results is observed here and the RMSE values between the simulations and the experimental data are smaller than those observed in section 4.2.4 with the maximum values presented in Table 4.3.

The test case presented here is a scenario where inaccuracies in the wake characterisation of the upstream turbine will be magnified further downstream. This is because the momentum sink term applied depends on the velocity at the down-

![Graphical representation of lateral velocity deficit and turbulence intensity profiles for the ADM-RANS simulations vs Mycek et al. at $x = 2D, 4D, 6D, 8D$ and $10D$ for the upstream turbine with $I_\infty = 15\%, C_t = 0.75$ and $u_\text{in} = 0.83 \text{ m s}^{-1}$. The same legend as in Fig. 4.6 is used.]

Figure 4.17: Lateral velocity deficit and turbulence intensity profiles for the ADM-RANS simulations vs Mycek et al. at $x = 2D, 4D, 6D, 8D$ and $10D$ for the upstream turbine with $I_\infty = 15\%, C_t = 0.75$ and $u_\text{in} = 0.83 \text{ m s}^{-1}$. The same legend as in Fig. 4.6 is used.
stream turbine location which lies within the wake of the upstream turbine. The Fluidity ADM-RANS model has accurately modelled the wake behind the upstream turbine which has in turn led to accurate wake characterisation of the downstream turbine. Once again, this has been achieved by correctly establishing the influential regions within the domain and refining the mesh therein, Fig. 4.16–4.17. This is an important result considering turbines are likely to be placed in the wake of upstream turbines in tidal turbine arrays in the real world.

Furthermore, the RANS models deal with the different ambient conditions effectively to produce a good match with the experimental results in both low and high background turbulence cases. It is interesting to see that, in both test cases considered in this study, the results from the $k-\omega$ SST model closely follow those produced via the $k-\omega$ model with correction terms. Given that the $k-\omega$ SST model does not require correction terms, it will be the RANS model of choice in the remainder of this work. Having said that, based on the results presented, there is

---

**Figure 4.18:** Lateral velocity deficit and turbulence intensity profiles for the ADM-RANS simulations vs Mycek et al. at $x = 2D$, $4D$, $6D$, $8D$ and $10D$ for the downstream turbine with $I_\infty = 3\%$, $C_t = 0.81$ and $u_{in} = 0.80 \text{ m s}^{-1}$. The same legend as in Fig. 4.6 is used.
Chapter 4: ADM-RANS validation: Flow past multiple scaled turbines

no disadvantage to using the $k - \omega$ RANS model, as long as appropriate correction terms are introduced to account for the short circuiting of the turbulence cascade.

Figure 4.19: Lateral velocity deficit and turbulence intensity profiles for the ADM-RANS simulations vs Mycek et al. at $x = 2D, 4D, 6D, 8D$ and $10D$ for the downstream turbine with $I_\infty = 15\%$, $C_t = 0.75$ and $u_{in} = 0.83 \text{ m s}^{-1}$. The same legend as in Fig. 4.6 is used.
### Table 4.3: Maximum RMSE values in velocity deficit and turbulence intensity comparing the simulation results to the experimental data. The corresponding wake profiles are shown in Fig. 4.16–4.19. The RMSE plots are shown in Fig. 4.20.

<table>
<thead>
<tr>
<th>Fluidity model</th>
<th>$1 - \frac{u_x}{u_0}$</th>
<th>I</th>
</tr>
</thead>
<tbody>
<tr>
<td>$k - \omega$</td>
<td>0.134</td>
<td>0.035</td>
</tr>
<tr>
<td>$k - \omega$ with correction</td>
<td>0.111</td>
<td>0.027</td>
</tr>
<tr>
<td>$k - \omega$ SST</td>
<td>0.081</td>
<td>0.026</td>
</tr>
</tbody>
</table>

**Figure 4.20:** RMSE values in velocity deficit and turbulence intensity for Fig. 4.16–4.19. The same legend as in Fig. 4.6 is used.
4.3.1 Viscosity

![Viscosity images](image)

**(a) $I_{\infty} = 3\%$**  
**(b) $I_{\infty} = 15\%$**

**Figure 4.21:** A 2D slice through the 3D domain showing the variation of viscosity ($\nu_{bg} + \nu_T$) for the $k-\omega$ (top), $k-\omega$ with correction terms (middle) and the $k-\omega$ SST (bottom) simulations.

![Turbulence length scale images](image)

**(a) $I_{\infty} = 3\%$**  
**(b) $I_{\infty} = 15\%$**

**Figure 4.22:** A 2D slice through the 3D domain showing the variation of turbulence length scale for the $k-\omega$ (top), $k-\omega$ with correction terms (middle) and the $k-\omega$ SST (bottom) simulations.
Fig. 4.21 displays 2D slices of the 3D domain at hub height showing the variation of viscosity within the domain for the various simulations of flow past a single turbine for the Mycek et al. test case. The results from the original $k - \omega$, the $k - \omega$ with correction terms and the $k - \omega$ SST model are shown. The variations of turbulence length scale, $l$, at hub height are also shown in Fig. 4.22.

The background viscosity has been set to a low value of $\nu_{bg} = 1 \times 10^{-6} \text{m}^2 \text{s}^{-1}$ throughout and therefore the variations observed in Fig. 4.21 are almost entirely due to variations in $\nu_T$. The inlet velocities are similar in both low and high $I_\infty$ cases, therefore in order to obtain the experimental $I_\infty$ values, different $k_{in}$ values were specified, Table 4.2. The inlet $\omega$ values determine the rate of decay of $k$, but unfortunately this was not measured experimentally and in this study $\omega_{in} = 0.5 \text{s}^{-1}$ was used for both low and high $I_\infty$. Given that $\nu_T$ is primarily determined by the ratio of $k$ to $\omega$, this has led to the different ambient viscosity values at low and high $I_\infty$ observed in Fig. 4.21. The inlet values for $k$, $\omega$ and hence $\nu_T$ used here for the $I_\infty = 15\%$ are very similar to those used by Shives and Crawford (2016), however therein a lower $\omega_{in}$ value was used for the $I_\infty = 3\%$ case. Nonetheless, similar to this study, therein the ambient $\nu_T$ was lower for the $I_\infty = 3\%$ case. Furthermore, the $k_{in}$ and $\omega_{in}$ values specified have led to a lower ambient $l$ value for the $I_\infty = 3\%$ case compared to the $I_\infty = 15\%$ case.

When using the original $k - \omega$ model the behaviour at low and high $I_\infty$ is very similar. In both cases $\nu_T$ increases in the wake downstream of the turbine and this is an indication that the bypass fluid is mixing in with the wake. This area of increased $\nu_T$ lasts longer at $I_\infty = 3\%$ due to the fact that the wake takes longer to re-energise for the lower $I_\infty$ case. However, this increase in viscosity leads to rapid wake re-energisation and the wake lengths obtained, for both low and high $I_\infty$, are shorter than those observed in the experiments, as demonstrated in Fig. 4.16–4.17. Furthermore, despite the similarities in the viscosity profiles, there are differences in the length scales observed, Fig. 4.22. At $I_\infty = 3\%$, the mixing in the wake leads to the formation of larger eddies, whereas at $I_\infty = 15\%$, no significant change in length scale is visible as a result of the presence of the turbine. Moreover, the model predicts no change in $l$ near the turbine in either low or high $I_\infty$.

With the $k - \omega$ with correction terms, the behaviour of $\nu_T$ is different and there
is distinguishable difference between the behaviour at low and high turbulence intensities. At $I = 3\%$, there is a drop in $\nu_T$ near the turbine due to the source terms introduced. As mentioned previously, these source terms are introduced in order to capture the short-circuiting of the turbulence cascade near the turbine. In the original $k - \omega$ model, there is minimal change in length scale, whereas with the introduction of the source terms, there is a noticeable drop in $l$ at the turbine. This represents the breakdown of the larger eddies to smaller scale and reduces the ability of the flow to mix over large length scales, which is represented as a local reduction in $\nu_T$. Further downstream, with the generation of $k$ due to the shear between the bypass flow and the wake, larger eddies are formed and the wake mixes with the bypass flow. This is represented by an increase in $\nu_T$ observed in the far wake. At $I = 15\%$, the behaviour is similar with a drop in $\nu_T$ and $l$ observed at the turbine. However, in this case the viscosity value in the wake does not increase beyond the ambient value and instead the viscosity everywhere is slightly reduced in the far wake and the profile is more dissipative compared to the one observed at $I = 3\%$. In both low and high $I_\infty$ cases, the local reduction in $\nu_T$ delays wake re-energisation and helps provide wake lengths that agree with experimental observations, Fig. 4.16–4.17.

In the $k - \omega$ SST simulations, the behaviour is similar to that observed in the $k - \omega$ with correction terms simulations. However, here the variation in $\nu_T$ near the turbine comes about due to the switch between $k - \varepsilon$ and $k - \omega$ near the turbine. Furthermore, in this case the region of reduced $\nu_T$ near the turbine is larger and more pronounced, but the far wake $\nu_T$ and $l$ profiles are similar to those of the $k - \omega$ with correction terms simulations, especially at $I = 15\%$. Hence, the velocity and $I$ profiles obtained are also very similar and both simulations provide a reasonable match with the experimental data, Fig. 4.16–4.17.
4.4 Conclusions

The validity and versatility of the Fluidity ADM-RANS model has been further assessed by reproducing two different published experimental flume tests. The ADM-RANS model does not take into account the swirl induced on the flow by the rotating blades, but it is still able to produce closely matching results by capturing the important turbulence characteristics in the far wake. Mesh optimisation techniques have been successfully employed to obtain an accurate representation of the turbine wakes whilst maintaining a relatively low computational cost and such techniques can be used in the future to aid larger scale simulations. Moreover, this has provided valuable insight into the importance of accounting for the short circuiting of the turbulence cascade due to the presence of the turbine when using RANS models. It has also been demonstrated that while the original \( k - \omega \) model is able to capture the short circuiting with the aid of additional correction terms, the \( k - \omega \) SST model is able to capture the far wake turbulence characteristics without the need to include correction terms that will require tuning. Having said that, correction terms may be beneficial to represent processes in the near wake of a tidal turbine that are not described by an ADM approach (Olczak et al., 2016).

The Fluidity ADM-RANS model performs admirably to capture the main flow features of interest. More importantly, it is able to do so without the need for excessive computational power and the methodology can be easily extended to model large arrays of tidal turbines. However, the major shortcoming is near-wall modelling due to the current bottom boundary representation. In the near-wall region, the flow velocity rapidly increases with distance from the wall and an extremely fine resolution is required to accurately model the steep velocity profiles present. This would significantly increase the computational cost of the simulations. Hence, the drag boundary condition has been used thus far to provide a minimalistic representation of the slower moving fluid near the bed. However, it has been demonstrated, both here and in chapter 3, that this boundary condition is inadequate in correctly modelling the bed generation of turbulence. Hence, in order to improve the ambient turbulence computation, wall functions need to be introduced to improve the bottom boundary representation of the current Fluidity model.
Chapter
FIVE

Investigating the accuracy of tidal turbine array optimisation

This chapter is derived from and expands upon a journal paper recently submitted to Renewable Energy by Abolghasemi et al. (2016a). The material therein has arisen as the result of the first author’s research and the co-authors have acted in guiding roles only.

5.1 Introduction

Currently many large-scale marine hydrodynamic models employed to study marine energy use the depth-averaged shallow water equations, rather than the full 3D Navier-Stokes equations. Gunn and Stock-Williams (2013) presented a comprehensive analysis of these models and assessed their ability to correctly predict tidal resources against measured ADCP data. Several site-specific numerical simulations have been documented. Sutherland et al. (2007) investigated the maximum tidal power potential of the Johnstone Strait, BC, Canada using a 2D finite element model (TIDE2D). Pham and Martin (2009) demonstrated how the Telemac-2D shallow water solver could be used to perform a tidal resource assessment at the Paimpol-Bréhat site in France and Martin-Short et al. (2015) used the Fluidity shallow water solver...
Chapter 5: Investigating the accuracy of tidal turbine array optimisation

to model arrays of tidal turbines in the Inner Sound of the Pentland Firth. In order
to numerically simulate arrays of turbines, these models usually adopt an approach
where the turbines are represented as a region of increased bottom drag (Sutherland et al., 2007; Divett et al., 2013; Martin-Short et al., 2015). Funke et al. (2014)
exploited the relatively low complexity of such an approach to improve turbine posi-
tions to maximise array power generation using iterative adjoint-based optimisation
techniques at a computational cost essentially independent of the number of tur-
bines.

A main shortcoming of the approach is that it can fail to account for important
turbulence physics and 3D effects, e.g. since the flow passing below and above the
turbine is not modelled. In a realistic environment the ambient turbulence intensity
has a significant effect on the structure of the turbine wake and its recovery (Black-
more et al., 2014; Mycek et al., 2014a,b), which is of course crucial in an array
design. However, when the depth-averaged shallow water equations are considered,
the resulting turbine wake structures are strongly dependent on the viscosity coeffi-
cient used and this is often set to a spuriously high value in order to ensure a stable
solution. The effect of ambient turbulence on the wake structure can therefore be
misrepresented if an inappropriate viscosity value is used. Turbulence models for
the shallow water equations have been suggested. For example, Nadaoka and Yagi
(1998) developed a model to simulate the evolution of horizontal large-scale eddies
in shallow water by characterising the turbulence as a coexistence of 3D turbulence,
with length scales smaller than water depth, and horizontal 2D eddies with much
larger length scales. Moreover, Mungar (2014) used a horizontal large eddy simula-
tion (HLES) in combination with a 3D $k-\epsilon$ model to examine the flow past tidal
turbines using Delft3D (Delft Hydraulics, 1999). However, for the steady flow sim-
ulations presented therein the influence of HLES on the velocities was negligible. It
was suggested that due to the implementation method of HLES in Delft3D, it is not
effective for steady flows without velocity fluctuations and therefore the influence of
HLES in combination with non-steady flows needs to be examined (Mungar, 2014).

Alternatively, the self-similar nature of far wake velocity profiles can be exploited
to predict the flow past arrays of tidal turbines. Recently, Stallard et al. (2015)
investigated the mean wake properties behind a single three-bladed scaled turbine
and demonstrated that for distances greater than 8 diameters downstream the ve-
locity deficit becomes two-dimensional and self-similar. Stansby and Stallard (2016) extended this to demonstrate that the superposition of self-similar velocity profiles can lead to accurate predictions of depth-averaged wake velocities downstream of arrays of tidal turbines. This allowed for computationally efficient optimisation of turbine positions for power generation. However, although the method was shown to be reliable for up to three rows, it may not capture large scale wake behaviour which would be generated by multiple rows (Stansby and Stallard, 2016).

Another alternative approach would be to use a higher-fidelity 3D model coupled with an appropriate 3D turbulence model. A number of these models have been developed and validated against experimental flume tests with promising results (Roc et al., 2013; Shives and Crawford, 2016; Olczak et al., 2016), but their high computational expense has generally prevented their application in large-scale regional simulations and within iterative design optimisation. However, mesh optimisation techniques have the potential to help bridge the gap and improve the accuracy of large-scale simulations without the need for excessive computational power (chapter 4).

In addition to this 2D vs 3D issue, Kramer and Piggott (2016) recently examined the depth-averaged approach and pointed out that the resulting force exerted on the flow by a parameterised turbine agrees well with the theoretical value for coarse mesh resolutions only. As the mesh size becomes smaller than the length scale of the wake recovery, the exerted force starts decreasing with decreasing mesh sizes. The reason for this lies in the fact that the assumption that the upstream velocity can be approximated by the local model velocity, is no longer valid. In order to resolve this issue, Kramer and Piggott (2016) suggested using ADM theory to derive a correction to the enhanced bottom drag formulation. This leads to an improved estimate of the usefully extractable energy.

In order to assess the suitability of using depth-averaged models to optimise turbine array configurations, in this chapter a depth-averaged model is used alongside a 3D hydrodynamic model based upon a RANS approach with resolved turbines using ADM theory. The sensitivity to the viscosity parameter employed in the depth-averaged model is investigated and consequently, with the aid of the ADM-RANS model, the viscosity value is tuned to improve the wake structure predicted
Chapter 5: Investigating the accuracy of tidal turbine array optimisation

by the lower fidelity model. Furthermore, the modifications suggested by Kramer and Piggott (2016) are used to improve the thrust and power predictions of the depth-averaged solution. The combination of improved wake characterisation and more accurate thrust and power calculations in the depth-averaged model are then used to optimise turbine positions in order to maximise the total extracted power from an array. This is performed within the OpenTidalFarm (OTF) framework, an open source software tool for simulating and optimising tidal turbine arrays developed by Funke et al. (2014). The depth-averaged results are then compared against 3D simulations of arrays of tidal turbines using the ADM-RANS model described in chapter 2, thus allowing for an investigation into the ultimate value of the depth-averaged adjoint-based optimisation used in OTF.

The chapter is organised as follows. First in section 5.2 the depth-averaged formulation used in OTF and the 3D Fluidity ADM-RANS model with mesh optimisation capabilities are introduced. Descriptions of the thrust and power calculations in each model in line with the modifications suggested by Kramer and Piggott (2016) are also presented. Then in section 5.3 the flow past a single turbine in an ideal channel is modelled using both models to determine their consistency and to arrive at a suitable viscosity value for use in OTF. This is then followed in section 5.4 by an examination of the flow past an array of 32 tidal turbines in an ideal channel where the turbine positions are optimised for maximum power. The flow solution and the power gain predicted by OTF is compared against the values obtained from the 3D Fluidity ADM-RANS model in order to provide insight into the accuracy of the depth-averaged simulations. Initially steady flow scenarios are considered, before extending the analysis to examine an unsteady flow scenario with a time-dependent inlet velocity. The chapter concludes with a general overview of the results.

5.2 Methodology

5.2.1 Depth-averaged model

OpenTidalFarm solves an optimisation problem constrained by the shallow water equations where the goal is to maximise power production $P$, i.e.
5.2: Methodology

\[
\max_{\mathbf{m}} \quad P(\mathbf{m}),
\]

subject to \(b_l \leq \mathbf{m} \leq b_u\), \(g(\mathbf{m}) \leq 0\),

where \(\mathbf{m}\) is a vector containing the turbine positions, bounds \(b_l \leq \mathbf{m} \leq b_u\) constrain the turbines to the (here rectangular) array area and the inequality constraint, \(g(\mathbf{m}) \leq 0\), enforces a minimum distance spacing constraint between adjacent turbines. In each optimisation iteration, a two-dimensional finite element shallow water model predicts the hydrodynamics for given forcing and turbine locations, and thus the performance of the current array configuration can be evaluated by computing the power produced. The gradient of the power extracted with respect to the turbine positions is then computed by solving the associated adjoint equations. These equations propagate causality backwards through the computation, from the power extracted back to the turbine positions. This yields the gradient at a cost almost independent of the number of turbines, which is crucial for any practical application targeted at large arrays (Funke et al., 2014). The optimisation is not limited to power production and can be used to maximise any functional of interest. More recently, Culley et al. (2016) extended the model to include economic costs and hence to optimise the turbine positions to maximise profit over the lifespan of the array.

In OTF the depth-averaged shallow water equations discretised are considered in the following form

\[
\frac{\partial \mathbf{u}}{\partial t} + \mathbf{u} \cdot \nabla \mathbf{u} - \nu \nabla^2 \mathbf{u} + \frac{c_b + c_t(\mathbf{m})}{H} ||\mathbf{u}|| \mathbf{u} = 0, \quad (5.3)
\]

\[
\frac{\partial \eta}{\partial t} + \nabla \cdot (H \mathbf{u}) = 0,
\]

where \(\mathbf{u}\) is the depth-averaged velocity, \(\nu\) is the kinematic eddy viscosity, \(\eta\) is the free surface displacement, \(H\) is the total water depth, \(g\) is the acceleration due to gravity, \(c_b\) and \(c_t(\mathbf{m})\) represent the background quadratic bottom friction and the local enhancement used to parameterise the presence of turbines, respectively. A turbine is modelled via an increased bottom friction over a small area representative of an individual turbine. This is achieved via a bump function which smoothly
increases the friction value at the turbine locations:

\[
\psi_{p,r}(x) = \begin{cases} 
  e^{1-1/(1-||\frac{x-p}{r}||^2)} & \text{for } ||\frac{x-p}{r}|| < 1, \\
  0 & \text{otherwise},
\end{cases}
\] (5.4)

where \(p\) and \(r\) are the centre and the support radius of a 1D bump function, respectively. A two-dimensional bump function is obtained by multiplying Eq. (5.4) by copies in both independent dimensions. The friction function of the \(i\)th turbine parameterised by friction coefficient \(K_i\) centred at point \((x_i, y_i)\) is then given by

\[
C_i(x, y) = K_i \psi_{x_i,r}(x) \psi_{y_i,r}(y).
\] (5.5)

The sum of the individual bottom friction fields associated with all \(N\) turbines is denoted as \(c_t(m)\) in equation (5.3) such that

\[
c_t(m) = \sum_{i=1}^{N} C_i.
\] (5.6)

5.2.2 3D ADM-RANS model

For the purpose of the 3D simulations, the Fluidity ADM-RANS model described in chapter 2 is used. The 3D ADM-RANS model is capable of dynamic mesh optimisation which is used to help reduce discretisation errors by refining the mesh in locations of numerical complexity or specific interest, e.g. regions with high velocity shear. This allows for better management of limited computational resources without having to compromise on the accuracy of the solution. This is a great advantage when simulating large arrays of tidal turbines. In the 3D ADM-RANS model, turbines are parametrised based on the ADM theory and turbulence is accounted for by incorporating turbulence models based on the RANS approach. A detailed description of the model has already been presented in chapter 2.

In order to set the appropriate loading on the disc, the Fluidity model uses the thrust coefficient, \(C_t\), to compute the magnitude of thrust loading that should be applied at the disc. This is uniformly spread across the volume of the disc and is implemented as a momentum sink term, Eq. (2.3). In the context of the numerical simulations,
it is important to ensure that the unperturbed upstream streamwise component of velocity, \( u_0 \), is predicted accurately and in the 3D model it is computed using

\[
u_t = \frac{1}{2} \left( 1 + \sqrt{1 - C_t} \right) u_0,
\]

(5.7)

where \( u_t \) is the average streamwise component of velocity over the elements making up the actuator disc (Hansen, 2000).

### 5.2.3 Thrust and power calculations

In order to ensure that the depth-averaged and fully 3D models are comparable, it is vital to ensure that firstly the same thrust is being applied in the different methods used to parameterise the presence of turbines, and secondly that the power calculation is consistent in both models. In the ADM-RANS model this is rather straightforward since the thrust applied is simply computed using

\[
T_{3d} = \frac{1}{2} \rho A_t C_t u_0^2,
\]

(5.8)

and by assuming that the turbine power is equal to the product of the applied thrust and the local velocity, the extracted power is determined to be

\[
P_{3d} = T_{3d} \times u_t.
\]

(5.9)

On the other hand, in OTF, the thrust applied is controlled by adjusting the amplitude of the bump function since

\[
T_{2d} = \int_{A_c} \rho c_t u_c^2,
\]

(5.10)

where \( c_t \) is the local friction coefficient enhancement used to parameterise the presence of turbines (5.6), \( u_c \) is the local depth-averaged velocity and \( A_c \) is the area enclosed by the bump function. A corresponding formula for the extracted power in OTF, \( P_{2d} \), is introduced later. Moreover, since (5.8) is expressed in terms of the unperturbed upstream velocity, \( u_0 \), in order to ensure consistency, a relationship between \( u_c \) and \( u_0 \) is required. This issue has been recently discussed by Kramer.
and Piggott (2016), a brief overview of which is provided here.

One of the main differences between the turbine representations in the ADM-RANS model and the depth-averaged model is that in the latter the turbine effectively blocks the entire channel depth. However, in the 3D ADM-RANS model, the horizontal and vertical components of velocity are resolved and the bypass flow passing underneath and above the turbine is also simulated. Hence, an equivalent thrust coefficient, $\hat{C}_t$, is defined for the depth-averaged model where

$$\hat{C}_t = \frac{A_t}{\hat{A}_t} C_t,$$  \hspace{1cm} (5.11)

with

$$\hat{A}_t = h \times w,$$

where $h$ is the depth of the channel at the turbine location and $w$ is the width of the bump function. Hence, analogous to (5.7), the following relationship can be used to express the depth-averaged $u_c$ in terms of $u_0$:

$$u_c = \frac{1}{2} \left( 1 + \sqrt{1 - \hat{C}_t} \right) u_0.$$  \hspace{1cm} (5.12)

Note that this is the same $u_0$ used in (2.3) and here the unperturbed 3D flow is assumed to be depth independent. Finally, equating the applied thrusts (5.10) to (5.8) and using (5.12) leads to

$$\int_{A_c} c_t = \frac{1}{2} A_t C_t \left( 1 + \sqrt{1 - \frac{4 A_t}{A_i} C_t} \right)^2.$$  \hspace{1cm} (5.13)

Therefore, by using (5.13) $c_t$ can be set such that it yields a thrust in the depth-averaged model that matches the thrust applied in the ADM-RANS model. Furthermore, similar to the ADM-RANS model, the extracted power is the product of the thrust and the velocity at the location of the turbine. Hence, the depth-averaged velocity $u_c$ needs to be converted to an equivalent local turbine velocity analogous
to $u_t$ used in (5.9). This can be achieved by combining (5.7) and (5.12) to yield

$$u_t = \frac{1 + \sqrt{1 - C_t}}{1 + \sqrt{1 - \hat{C}_t}} u_c, \quad (5.14)$$

and this can be used to compute the extracted power using

$$P_{2d} = T_{2d} \times u_t. \quad (5.15)$$

### 5.3 Viscosity sensitivity

The ability to correctly account for the wake behind each turbine is of utmost importance when modelling arrays of tidal turbines. This was the main motivation behind the RANS approach adopted in the 3D Fluidity ADM-RANS model as the $k - \omega$ SST model used helps determine the wake length and momentum recovery depending on the ambient turbulence conditions (chapter 4). On the other hand, in OTF the wake length can at present only be controlled via an eddy viscosity coefficient which in the current version of the software takes the same value everywhere in the domain. The reason for this lies in the fact that in order to keep iterative based turbine optimisation feasible, the computational cost behind each flow solve must be kept to a minimum and this limitation has not allowed for the inclusion of turbulence models in OTF thus far. However, by careful calibration of the viscosity coefficient, the disadvantages of not modelling for turbulence directly can be better understood and therefore minimised.

In order to determine the significance of the viscosity coefficient in OTF a sensitivity study was carried out. Prior to proceeding with tidal turbine array simulations, the flow past a single turbine in an idealised channel is modelled using both the 3D Fluidity ADM-RANS model and the depth-averaged OTF package. The domain considered for this scenario is illustrated in Fig. 5.1.
Chapter 5: Investigating the accuracy of tidal turbine array optimisation

Figure 5.1: Numerical domain with turbine diameter $D = 18.75\, \text{m}$ and thickness $D/4$ used in the 3D ADM-RANS simulations. The dark grey area represents the actuator disc positioned at mid-depth. In the OTF simulations an equivalent 2D depth-averaged domain was used with a $D \times D$ square turbine drag area.

5.3.1 Boundary conditions

In the ADM-RANS model, at the inlet, a Dirichlet boundary condition with constant inlet values, $u_{\text{in}}$, $k_{\text{in}}$, and $\omega_{\text{in}}$ is applied, and the side walls and top surface are set to free slip. Furthermore, a zero flux boundary condition has been applied at the walls for both $k$ and $\omega$ and a zero pressure outflow boundary condition has been applied at the outlet. In order to simulate the flow inside the channel, it is important to capture the vertical asymmetry caused by the slower moving fluid near the bed. Hence, in the velocity field a quadratic drag boundary condition, with a non-dimensional drag coefficient of $C_D = 0.0025$, is applied to the bottom surface. For more detail on the quadratic drag boundary condition used in the ADM-RANS model, the reader is referred to section 3.7.1.

Similarly, in OTF, a Dirichlet velocity boundary condition with a constant inlet value is applied with the side walls set to free slip. The free surface displacement is set to zero at the outlet and a quadratic bottom friction $c_b = 0.0025$ is prescribed. This drag is effectively applied across the whole depth in the depth-averaged model, whereas the quadratic drag specified in the 3D ADM-RANS model is only applied at the bottom surface.

Fig. 5.2 illustrates the variation of $\nu_T$ at mid-depth in the 3D ADM-RANS model for the flow past a turbine with $u_{\text{in}} = 2.9\, \text{m}\, \text{s}^{-1}$, $k_{\text{in}} = 0.126\, \text{m}^2\, \text{s}^{-2}$, $\omega_{\text{in}} = 0.1\, \text{s}^{-1}$ and $C_t = 0.85$. Generally, the $\nu_T$ value is lower inside the wake downstream of the turbine. Furthermore, the smallest values are observed immediately upstream of the turbine and in the region between the slow moving fluid in the near wake and the accelerated bypass flow. These regions inherit the greatest levels of shear and therefore the ADM-RANS model has adjusted the $k$ and $\omega$ values such that $\nu_T$ is
5.3: Viscosity sensitivity

Figure 5.2: Variation of $\nu_T$ at mid-depth for the 3D ADM-RANS model with the domain used shown in Fig. 5.1.

reduced at these locations. If the high $\nu_T$ present at the inlet was maintained everywhere, this would have encouraged more mixing and a shorter wake. Therefore, by reducing $\nu_T$ locally, the ADM-RANS model has delayed wake re-energisation. It was demonstrated in chapters 3 and 4 that this approach can help predict correct wake lengths for different ambient turbulence values which agree with experimental observations. Note that this variation in eddy viscosity is not present in the depth-averaged OTF model since a constant value is used there.

In order to accurately understand the consequences of using different viscosity coefficients, it was decided to compare the results from the Fluidity ADM-RANS model against those obtained via the depth-averaged OTF package for a range of different inlet conditions. This is crucial given the periodic nature of tidal flows which are of interest here. Therefore, a series of simulations with varying inlet velocities were carried out ranging from 0.9 m s$^{-1}$ to 3.9 m s$^{-1}$ using both models.

The effect of turbulence and how the turbulence intensity ($I$) might change with varying velocities is also critical and therefore three different scenarios have been considered in the ADM-RANS simulations:

(a) as $u_{in}$ increases, both $k_{in}$ and $\omega_{in}$ are unaffected and therefore $\nu_T$ is unaffected but $I$ decreases

(b) as $u_{in}$ increases, $k_{in}$ also increases but $\omega_{in}$ is unaffected and therefore $I$ is
unaffected but $\nu_T$ increases

(c) as $u_{in}$ increases, both $k_{in}$ and $\omega_{in}$ also increase and therefore both $I$ and $\nu_T$ are unaffected

Initially the inlet values are set to $u_{in} = 2.9 \text{ m s}^{-1}$, $k_{in} = 0.126 \text{ m}^2 \text{s}^{-2}$ and $\omega_{in} = 0.1 \text{ s}^{-1}$. This corresponds to $I = 10\%$ and a turbulence length scale ($l$) of 39.4 m, which is slightly larger than the channel depth. Consequently, as $u_{in}$ is varied, $k_{in}$ and $\omega_{in}$ are, if necessary, adjusted relative to these values and in-line with the scenarios described above. This leads to various inlet conditions for scenarios (a)–(c) and the resulting inlet values are shown in Table 5.1–5.3. The $\nu_T$, $I$ and $l$ presented have been computed using

\begin{align*}
I &= \sqrt{\frac{2}{3}} \frac{k_{in}}{u_{in}}, \\
\nu_T &= \frac{k_{in}}{\omega_{in}}, \\
l &= \sqrt{K_{in}} \frac{\beta^* \omega_{in}}{\omega_{in}},
\end{align*}

with non-dimensional coefficient $\beta^* = 0.09$ (Menter et al., 2003).

Out of the three scenarios considered in the ADM-RANS simulations, scenario (c) appears to be the most realistic one. The reason for this lies in the fact that changes in the velocity fields will naturally alter the horizontal and vertical shear profiles within the flow and in order to capture this behaviour both $k$ and $\omega$ fields will have to be modified. Furthermore, it has been shown that increasing ambient $I$ leads to faster wake re-energisation (Mycek et al., 2014a,b) and the importance of viscosity on the rate of wake re-energisation has already been stressed. Therefore, in scenario (c), $k_{in}$ and $\omega_{in}$ both increase with increasing $u_{in}$ in order to maintain the same $I$ and $\nu_T$ at the inlet. Hence, $\nu_T$ at the inlet only changes if $I$ at the inlet changes and this reflects the close relationship between the two. However, one of the issues with scenario (c) is that, as a result of fixing $\nu_T$ at the inlet, the inlet $l$ values drop with increasing $u_{in}$ and this is not physical. Alternatively, a scenario can be envisaged where $I$ and $l$ remain constant with increasing $u_{in}$, but in that case $\nu_T$ at the inlet will no longer remain constant. Generally, at a realistic tidal site the change in flow
features will be more complicated than the constant turbulent intensity assumption used here and the variations in $k$ and $\omega$ will depend on the particular site under consideration.

<table>
<thead>
<tr>
<th>$u_{in}$ (m s$^{-1}$)</th>
<th>$k_{in}$ (m$^2$ s$^{-2}$)</th>
<th>$\omega_{in}$ (s$^{-1}$)</th>
<th>$I$</th>
<th>$\nu_T$ (m$^2$ s$^{-1}$)</th>
<th>$l$ (m)</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.9</td>
<td>0.126</td>
<td>0.1</td>
<td>0.32</td>
<td>1.26</td>
<td>39.4</td>
</tr>
<tr>
<td>1.9</td>
<td>0.126</td>
<td>0.1</td>
<td>0.15</td>
<td>1.26</td>
<td>39.4</td>
</tr>
<tr>
<td>2.9</td>
<td>0.126</td>
<td>0.1</td>
<td>0.10</td>
<td>1.26</td>
<td>39.4</td>
</tr>
<tr>
<td>3.9</td>
<td>0.126</td>
<td>0.1</td>
<td>0.07</td>
<td>1.26</td>
<td>39.4</td>
</tr>
</tbody>
</table>

Table 5.1: Different inlet conditions for scenario (a) of the ADM-RANS simulations.

<table>
<thead>
<tr>
<th>$u_{in}$ (m s$^{-1}$)</th>
<th>$k_{in}$ (m$^2$ s$^{-2}$)</th>
<th>$\omega_{in}$ (s$^{-1}$)</th>
<th>$I$</th>
<th>$\nu_T$ (m$^2$ s$^{-1}$)</th>
<th>$l$ (m)</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.9</td>
<td>0.012</td>
<td>0.1</td>
<td>0.10</td>
<td>0.12</td>
<td>12.2</td>
</tr>
<tr>
<td>1.9</td>
<td>0.054</td>
<td>0.1</td>
<td>0.10</td>
<td>0.54</td>
<td>25.8</td>
</tr>
<tr>
<td>2.9</td>
<td>0.126</td>
<td>0.1</td>
<td>0.10</td>
<td>1.26</td>
<td>39.4</td>
</tr>
<tr>
<td>3.9</td>
<td>0.228</td>
<td>0.1</td>
<td>0.10</td>
<td>2.28</td>
<td>53.1</td>
</tr>
</tbody>
</table>

Table 5.2: Different inlet conditions for scenario (b) of the ADM-RANS simulations.

<table>
<thead>
<tr>
<th>$u_{in}$ (m s$^{-1}$)</th>
<th>$k_{in}$ (m$^2$ s$^{-2}$)</th>
<th>$\omega_{in}$ (s$^{-1}$)</th>
<th>$I$</th>
<th>$\nu_T$ (m$^2$ s$^{-1}$)</th>
<th>$l$ (m)</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.9</td>
<td>0.012</td>
<td>0.0095</td>
<td>0.10</td>
<td>1.26</td>
<td>128.1</td>
</tr>
<tr>
<td>1.9</td>
<td>0.054</td>
<td>0.043</td>
<td>0.10</td>
<td>1.26</td>
<td>60.0</td>
</tr>
<tr>
<td>2.9</td>
<td>0.126</td>
<td>0.1</td>
<td>0.10</td>
<td>1.26</td>
<td>39.4</td>
</tr>
<tr>
<td>3.9</td>
<td>0.228</td>
<td>0.181</td>
<td>0.10</td>
<td>1.26</td>
<td>29.3</td>
</tr>
</tbody>
</table>

Table 5.3: Different inlet conditions for scenario (c) of the ADM-RANS simulations.
Chapter 5: Investigating the accuracy of tidal turbine array optimisation

5.3.2 Results

In the 3D ADM-RANS model a turbine with $C_t = 0.85$ and $D = 18.75$ m has been assumed. In OTF the turbine cell area, $A_c$, where the bump function is defined is a square region with side $D$ explicitly resolved by multiple triangles in the unstructured mesh. In order to ensure the same thrust is applied in both models, Eq. (5.13) is used to compute the equivalent OTF parameter to be $\int_{A_c} c_t = 0.418$. Consequently, this requires $K = 1.147$ in Eq. (5.5). In the OTF simulations a range of different eddy viscosity values ranging from $0.1 \text{ m}^2\text{s}^{-1}$ to $1 \text{ m}^2\text{s}^{-1}$ have been considered in increments of $0.1 \text{ m}^2\text{s}^{-1}$. This will help outline the limitations of using the same viscosity value everywhere for the different scenarios considered. This range of eddy viscosity values has been chosen since it is similar to the range of $\nu_T$ values observed in the ADM-RANS simulations, Fig. 5.2.

In the 3D ADM-RANS simulations, mesh optimisation has been used to refine the mesh in regions of high curvature in the velocity, $k$, and $\omega$ fields. The minimum and maximum values of the element edge length were set to $l_e/D = 0.125$ and $l_e/D = 2$, respectively. A timestep of 2 s was used and the mesh was optimised after every 10 timesteps. The results from the 3D ADM-RANS simulations are then depth-averaged in a post-processing step to allow for a comparison against the OTF results. In OTF, an unstructured fixed 2D mesh has been used with minimum $l_e/D = 0.125$ near the turbine and maximum $l_e/D = 1.07$ at the boundaries. The minimum value used is the same as the one used in the ADM-RANS model and the maximum value is almost half the value used in the 3D model.

Fig. 5.3 displays lateral velocity deficit profiles at $6D$ and $12D$ downstream of the turbine for the various OTF and ADM-RANS simulations at four different $u_{in}$ values. The dotted lines represent the OTF results for different eddy viscosity values. As $\nu$ increases from $0.1 \text{ m}^2\text{s}^{-1}$ to $1 \text{ m}^2\text{s}^{-1}$, the velocity deficit in the wake is reduced and the lateral expansion of the wake increases as expected. Furthermore, for the same $\nu$ value in OTF, the wake length grows as $u_{in}$ increases from $0.9 \text{ m} \text{s}^{-1}$ to $3.9 \text{ m} \text{s}^{-1}$. This is reflected in the higher velocity deficit values observed. Also note that the lateral width of the wake decreases with increasing $u_{in}$. This is due to the fact that as $u_{in}$ increases the difference in velocity between the bypass flow and the wake also increases and this delays wake re-energisation.
5.3: Viscosity sensitivity

Figure 5.3: Lateral velocity deficit profiles at 6D and 12D at four different $u_0$ values for the OTF and the Fluidity ADM-RANS simulations. The dotted lines represent the OTF results where $\nu$ is increased from $0.1 \text{ m}^2\text{s}^{-1}$ to $1 \text{ m}^2\text{s}^{-1}$. The 3D ADM-RANS results have been depth-averaged and are plotted on top of the OTF results to help identify the OTF $\nu$ value that best matches the ADM-RANS profiles.
The results from the three scenarios considered using the ADM-RANS model have been depth-averaged and are plotted on top of the OTF results, Fig. 5.3. This will help contrast the OTF results against those obtained using the 3D ADM-RANS model. The three different scenarios considered in the ADM simulations lead to significantly different results. In scenarios (a) and (c) the behaviour is similar to the OTF results in that the wake lengths grow with increasing $u_{in}$. This growth is more pronounced in scenario (a) than in scenario (c), where the wake lengths are not significantly affected by the increase in $u_{in}$. In scenario (b) however, the opposite can be observed where the deficit decreases with increasing $u_{in}$. Scenario (b) is the only scenario where the inlet $\nu_T$ is changing with $u_{in}$, in the other two scenarios $\nu_T$ at the inlet is fixed. Hence, compared to the other two scenarios, the inlet $\nu_T$ is lower at $u_{in} = 0.9 \text{ m s}^{-1}$. This indicates less mixing between the bypass flow and the wake, which leads to longer wakes and greater velocity deficit values. As $u_{in}$ increases to $3.9 \text{ m s}^{-1}$, the inlet $\nu_T$ also increases and this encourages more mixing and shorter wake profiles. Furthermore, although the inlet $\nu_T$ value of $1.26 \text{ m}^2\text{s}^{-1}$ present in the 3D ADM-RANS simulations is outside the range of $\nu$ values considered in OTF, it has been shown that a value of $1 \text{ m}^2\text{s}^{-1}$ overpredicts the rate of wake re-energisation, Fig. 5.3, and increasing the OTF $\nu$ value further will only lead to a larger discrepancy between the two models.

Overall, Fig. 5.3 highlights the shortcomings that result due to the fixed viscosity constraint in the OTF simulations given that there is not an OTF setup (i.e. fixed $\nu$ value) that agrees with the ADM-RANS profiles perfectly. This suggests that even if a constant eddy viscosity is to be used everywhere in the domain, this value should be at least adjusted in-line with the upstream velocity value or the point in the tidal cycle. In order to shed light on the relationship between $u_{in}$ and the most suitable viscosity value, the OTF viscosity value that leads to the best match velocity deficit profile for each ADM-RANS run has been selected and the results are presented in Fig. 5.4.

In order to come up with a simple relationship between $u_{in}$ and the most appropriate $\nu$ (recalling that only $0.1 \text{ m}^2\text{s}^{-1}$ increments in values were considered) to be used in OTF, scenario (c) is the one considered as it is the most realistic scenario. Hence, the following linear relationship is suggested where

\[ \text{In order to come up with a simple relationship between} \ u_{in} \ \text{and the most appropriate} \ \nu \ \text{(recalling that only} \ 0.1 \text{ m}^2\text{s}^{-1} \ \text{increments in values were considered) to be used in OTF, scenario (c) is the one considered as it is the most realistic scenario. Hence, the following linear relationship is suggested where} \]
5.4 Channel flow

Having established the means to address the key differences in the wake profiles predicted by the depth-averaged OTF model and the 3D ADM-RANS model, the flow past an array of 32 tidal turbines was considered using both models in order to assess the suitability of the adjoint-based optimisation used in OTF. Initially, steady flow cases are considered with constant inlet velocities, and then an unsteady case is presented with a time-dependent inlet velocity.
Chapter 5: Investigating the accuracy of tidal turbine array optimisation

5.4.1 Steady flow

The same ideal channel of the previous section is used here with the same boundary conditions described in section 5.3.1. Two different inlet velocity values of 1.9 m s\(^{-1}\) and 2.9 m s\(^{-1}\) have been considered to allow for a comprehensive comparison between the two models. In this section, for the depth-averaged OTF simulations the steady state shallow water equations are solved and for the 3D ADM-RANS simulations the unsteady RANS equations are run until steady state is achieved.

5.4.1.1 OpenTidalFarm – depth-averaged

Initially the 32 turbines were arranged in 9 rows in a staggered layout and OTF was used to maximise the extracted power from the array by optimising the turbine positions for maximum power. The eddy viscosity coefficient used has been calculated using Eq. (5.19) and therefore \(\nu = 0.435 \text{ m}^2\text{s}^{-1}\) and \(\nu = 0.585 \text{ m}^2\text{s}^{-1}\) for \(u_{\text{in}} = 1.9 \text{ m s}^{-1}\) and \(u_{\text{in}} = 2.9 \text{ m s}^{-1}\), respectively. In these simulations the 32 turbines are assumed to be identical and the turbine properties (i.e. \(A_c\) and \(K\)) are the same as those used in section 5.3.2. The turbines were constrained to a 750 m × 375 m rectangular area in the middle of the channel as illustrated in Fig. 5.5. The OTF package has been used to optimise the position of the turbines within this area in order to maximise the power output of the array of 32 turbines in each case. A minimum spacing of 2\(D\) between adjacent turbines was enforced in the OTF optimisation.

The optimisation results are shown in Fig. 5.6–5.7 with the initial staggered turbine layout on the left and the optimised turbine layout on the right. The total array extracted power has been computed using Eq. (5.15) and the results are shown in Table 5.4. In both cases the optimised layout leads to substantial increases in total power with a 32\% increase for the \(u_{\text{in}} = 1.9 \text{ m s}^{-1}\) case and a 37\% increase for the \(u_{\text{in}} = 2.9 \text{ m s}^{-1}\) case. The two optimised layouts are very similar and in both cases OTF has moved the turbines, mainly in the lateral direction, in order to make sure no turbine is placed in the wake of an upstream turbine. This ensures a higher upstream velocity, which will in turn result in greater extracted power by the turbine.
5.4: Channel flow

![Diagram of channel flow with dimensions and turbine positions]

**Figure 5.5:** OTF domain for an array of 32 turbines in an ideal channel where the grey area represents the site where turbines can be positioned.

**Figure 5.6:** OTF results showing the initial staggered layout (12.98 MW) on the left and the optimised layout (17.12 MW) on the right for the $u_{in} = 1.9 \text{ m/s}^{-1}$ case.

### 5.4.1.2 Fluidity – 3D ADM-RANS

The Fluidity ADM-RANS model has been used to check the power predictions of OTF and the suitability of the eddy viscosity values used in the simulations of the array of 32 turbines. Hence, the flow past the initial staggered layouts, the final optimised layouts as well as two randomly chosen intermediate layouts, have been simulated using the 3D ADM-RANS model. The inlet turbulence properties have been set using scenario (c), described previously in section 5.3.1. As with the OTF simulations, the 32 turbines are assumed to be identical with $C_t = 0.85$. Note that $C_t$ is assumed to be constant in this work; however thrust curves, where $C_t$ is ex-
pressed as a function of $u_0$, could also be easily used instead. This represents the performance of a real device more accurately as it enables the model to take into account the cut-in speed below which the turbine does not operate and the rated speed above which $C_t$ decreases to maintain a constant power yield (Martin-Short et al., 2015). This will make the model better suited for the simulation of turbines during operation and will be necessary when simulating realistic scenarios.

The mesh optimisation capabilities of the Fluidity ADM-RANS model makes these simulations feasible since there is no need to produce a fine mesh prior to the simulations, anticipating the location of the wakes (which in a later solution vary both in space as well as time). This is especially true when considering the optimised layouts, where the turbine layout is irregular. Once again the mesh is refined in regions of high curvature in the velocity, $k$, and $\omega$ fields and the edge lengths values and frequency of adapts used were identical to those described in section 5.3.2.

Fig. 5.8 illustrates the wakes formed behind the turbines in both the staggered and the optimised layouts along with a 2D slice through the 3D domain showing the optimised mesh. The total array extracted powers computed using Eq. (5.9) are compared against the values obtained from the OTF simulations and the results
5.4: Channel flow

(a) staggered layout (46.48 MW)

(b) optimised layout (59.12 MW)

Figure 5.8: Iso-surfaces of constant speed, $||\mathbf{u}|| = 0.85 \, u_{in}$, are shown here to illustrate the wakes formed behind the turbine array in both the staggered and optimised layouts for the Fluidity ADM-RANS simulations with $u_{in} = 2.9 \, \text{m s}^{-1}$. A 2D slice at hub height through the 3D domain is also shown to demonstrate the optimised mesh where a higher resolution is used near the turbines and their wakes. The turbine colours correspond to the relative power output with red being the highest and blue corresponding to the lowest.

are presented in Table 5.4 and Fig. 5.9. The power predictions of the two models generally agree well with each other with a 8.5% difference in the worst case. This difference is by no means insignificant, but it gives confidence that there is robustness in the improved array designs of OTF yielding increased power.
Table 5.4: Steady flow simulation array power values comparing the variable viscosity OTF model against the 3D ADM-RANS model.

<table>
<thead>
<tr>
<th>$u_{in}$</th>
<th>layout</th>
<th>OTF power</th>
<th>ADM-RANS power</th>
<th>difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.9 m s$^{-1}$</td>
<td>staggered</td>
<td>12.98 MW</td>
<td>14.18 MW</td>
<td>-8.5%</td>
</tr>
<tr>
<td>1.9 m s$^{-1}$</td>
<td>optimised</td>
<td>17.17 MW</td>
<td>17.66 MW</td>
<td>-2.8%</td>
</tr>
<tr>
<td>2.9 m s$^{-1}$</td>
<td>staggered</td>
<td>45.48 MW</td>
<td>46.48 MW</td>
<td>-2.2%</td>
</tr>
<tr>
<td>2.9 m s$^{-1}$</td>
<td>optimised</td>
<td>62.13 MW</td>
<td>59.12 MW</td>
<td>+5.1%</td>
</tr>
</tbody>
</table>

In order to further demonstrate the importance of using an appropriate viscosity value in the depth-averaged model, the OTF simulations were also run with a higher eddy viscosity value of $\nu = 3$ m$^2$s$^{-1}$, as used in Funke et al. (2014). The resulting initial staggered array power outputs were 16.90 MW and 57.87 MW for $u_{in} = 1.9$ m s$^{-1}$ and $u_{in} = 2.9$ m s$^{-1}$, respectively. These values are 19.2% ($u_{in} = 1.9$ m s$^{-1}$) and 24.5% ($u_{in} = 2.9$ m s$^{-1}$) greater than the values predicted by the ADM-RANS model. The reason for this lies in the fact that the higher viscosity used in OTF leads to faster wake recoveries which in turn results in greater velocities at the turbine locations. Given that the power output scales with velocity cubed, small differences in velocity are magnified in the power output values and this has led to exaggerated array power output values in OTF. This highlights the importance of correctly calibrating viscosity in the depth averaged model as small differences in $\nu$ will result in significant errors in the predicted array power outputs.
5.4.2 Power per turbine

In order to examine the discrepancies between the two models, a closer examination of the extracted power values is presented in this section. A comparison between the power output per turbine predicted by the two models for both upstream velocity values is presented in Fig. 5.10. In this plot the power per turbine predicted by the 3D ADM-RANS model, Eq. (5.9), is plotted against the value predicted by the OTF simulations, Eq. (5.15). The results for both the initial staggered layouts and the final optimised layouts are shown. The diagonal line indicates a perfect match between the two models and therefore the results are judged based on how much they deviate away from this line.

The OTF model has predominantly underpredicted the power values for the staggered layout with maximum differences of 20% for the $u_{in} = 1.9 \text{ m s}^{-1}$ case and 15% for the $u_{in} = 2.9 \text{ m s}^{-1}$ case. In fact, the differences between OTF and ADM-RANS are largest for the turbines with the lowest power outputs since these turbines are the ones located in the final row of the staggered layout. The reason for this lies in the fact that for the first row since there are no turbines upstream, inaccuracies in the wake profile predictions do not affect the power predictions. However, further downstream in the later rows, the discrepancies between the two models grow due to the limitations associated with the wake profile predictions of OTF. These inaccuracies are amplified and therefore the worst agreements can be observed in the final row of the staggered layout. This is not surprising given that the velocity values at these locations are heavily dependent on the values upstream and therefore small discrepancies upstream would be amplified at these locations.

The results for the optimised layout are more balanced and the maximum differences are 6% for the $u_{in} = 1.9 \text{ m s}^{-1}$ case and 12% for the $u_{in} = 2.9 \text{ m s}^{-1}$ case. In these layouts, the turbines are not positioned in the wake of any upstream turbine and therefore inaccuracies in wake profile predictions of OTF are not magnified as much. Overall, given the assumptions used in this study and the different approaches used to model the turbines, a good agreement between the power values predicted by the two models can be observed. This is also reflected in the relatively small differences observed in the predicted total array power outputs, Table 5.4.
Chapter 5: Investigating the accuracy of tidal turbine array optimisation

5.4.3 Unsteady flow

Having shown good qualitative agreement between the wake profiles and the power production predicted by the two models for constant inlet velocities, in this section the analysis is extended to examine an unsteady scenario with a time dependent inlet velocity. It is vital to be able to extend the analysis to time-dependent scenarios given that this is a closer representation of real tidal cycles and that the wakes generated during one cycle are likely to influence the wake generation during subsequent cycles. Once again, the same ideal channel described in Fig. 5.5 is also used here. However, here the inlet velocity is modified to follow a sinusoidal profile such that

$$u_{\text{in}}(t) = u_{\text{max}} \sin(2\pi(t/\tau)),$$

where $\tau = 12$ h and $u_{\text{max}} = 3.9$ m s$^{-1}$. A half-cycle is considered here and therefore flow reversal is not modelled. Moreover, the boundary conditions, apart from at the inlet, are identical to those described in section 5.3.1.

Figure 5.10: OTF vs. Fluidity ADM-RANS power per turbine comparison plot for the staggered and optimised layouts. The diagonal line indicates a perfect match between the two models.
5.4: Channel flow

5.4.3.1 OpenTidalFarm – depth-averaged

Previously, in the steady flow OTF scenarios considered, the array power output was maximised for the particular inlet velocity value. However, for unsteady simulations, OTF optimises the turbine positions in order to maximise the array power integral over time. Hence, the optimised positions reflect the array layout that yields the maximum power integral for the duration of the 6 h half-cycle. Furthermore, in the OTF simulations presented here, the eddy viscosity coefficient is also time dependent and is set to vary with the inlet velocity according to the relationship derived in section 5.3, Eq. (5.19). This will further examine the benefits of varying the viscosity in-line with the inlet velocity, as opposed to using a constant spurious viscosity value throughout.

Fig. 5.11 displays the optimisation results with the initial staggered turbine layout on the left and the optimised turbine layout on the right. As in section 5.4.1, a minimum spacing of $2D$ between adjacent turbines was enforced in the OTF optimisation. As might be expected in this scenario without flow reversal, the optimised layout is very similar to the two optimised layouts observed for the steady flow cases considered, Fig. 5.6–5.7. Once again, OTF has avoided placing turbines downstream of each other as far as possible. Here, OTF predicts a power integral of 281 MW h for the initial staggered layout which increases to 386 MW h with the optimised layout. This is a substantial increase of 37%.

![Figure 5.11: OTF results showing the initial staggered layout on the left and the optimised layout on the right for the time-dependent inflow velocity case.](image)
5.4.3.2 Fluidity – 3D ADM-RANS

The 3D ADM-RANS model was again used to assess the OTF power predictions. In the ADM-RANS model, the turbulence properties (i.e. $k$ and $\omega$) were also assumed to be time dependent and follow scenario (c), described in section 5.3, where $I$ and $\nu_T$ are unaffected by changes to the inlet velocity. The variation of inlet velocity, $k$ and $\omega$ with time is shown in Fig. 5.12. One of the issues with scenario (c) is that the turbulence length scale becomes very large for small values of $k$ and $\omega$. Therefore, minimum values of $k_{\text{min}} = 1.80 \times 10^{-3} \text{m}^2 \text{s}^{-2}$ and $\omega_{\text{min}} = 1.43 \times 10^{-3} \text{s}^{-1}$ have been specified to ensure that the turbulence length scale is capped at $l_{\text{max}} = 330 \text{m}$ which is equivalent to 10 times the depth of the channel. This value is also less than half the channel width and it is not unreasonable to assume the existence of horizontal eddies with this length scale within the flow. The variation of $l$ at the inlet over time is also shown in Fig. 5.12.

![Graphs showing variation of boundary conditions with time](image)

**Figure 5.12:** Variation of the inlet boundary conditions with time for the unsteady flow ADM-RANS simulations during the 6 h half-cycle.

Once again, mesh optimisation has been used with the same criteria described previously in section 5.3. The mesh optimisation capabilities available in the ADM-RANS model help reduce the computational cost of these simulations. A high resolution initial mesh that anticipates the position and extent of the wakes is not required. Instead, the initial mesh simply needs to resolve the turbines (i.e. actuator discs) with sufficient resolution, saving time on the pre-processing. Thereafter, the model will, if and when necessary, increase the mesh resolution downstream of the turbines.
as the wakes start to develop. Thus, capturing the wake interactions with sufficient accuracy. In order to demonstrate this, 2D slices at hub height through the 3D domain showing the optimised mesh at various times of the unsteady simulation past the optimised layout are shown in Fig. 5.13. As demonstrated, the number of elements used increases with increasing $u_{in}$ in order to ensure that the wakes downstream of the turbines are correctly captured.

![Figure 5.13](image)

**Figure 5.13:** 2D slices at hub height through the 3D mesh at various times of the ADM-RANS unsteady flow simulation past the optimised layout.

The 3D ADM-RANS model was used to simulate the time dependent flow past the initial staggered and the final optimised layouts. Fig. 5.14 shows the total array power production values predicted by the ADM-RANS model, as well as OTF, for both layouts. OTF depth-averaged simulations were also run with $\nu = 0.1 \text{ m}^2\text{s}^{-1}$ and $\nu = 1 \text{ m}^2\text{s}^{-1}$ corresponding to the lower and upper bound eddy viscosity values considered in section 5.3. Moreover, a spurious high eddy viscosity value of $\nu = 3 \text{ m}^2\text{s}^{-1}$, as was used in Funke et al. (2014), was also considered. The various power integral values obtained for the two layouts shown in Fig. 5.11 are presented in Table 5.5.
As demonstrated earlier in section 5.3, increasing $\nu$ leads to shorter wakes in OTF. In the staggered case the turbines are positioned in the wake of upstream turbines. Hence, shorter wakes leads to higher velocities at downstream turbine locations and this results in greater array power extraction values. Therefore, the OTF run with constant $\nu = 3 \, \text{m}^2 \, \text{s}^{-1}$ predicts the highest array power integral and overestimates the ADM-RANS value by more than 27% (Table 5.5). On the other hand, the OTF run with constant $\nu = 0.1 \, \text{m}^2 \, \text{s}^{-1}$ predicts the lowest array power integral and underestimates the ADM-RANS value by 22%. The closest match using the constant eddy viscosity OTF model is achieved with $\nu = 1 \, \text{m}^2 \, \text{s}^{-1}$ which overpredicts the power integral by almost 9%. The best agreement between the depth-averaged OTF and the 3D ADM-RANS results is achieved using the variable eddy viscosity OTF simulation, Fig. 5.14a, which overpredicts the power integral by only 1%.

<table>
<thead>
<tr>
<th>Layout</th>
<th>Staggered (MW h)</th>
<th>Optimised (MW h)</th>
</tr>
</thead>
<tbody>
<tr>
<td>ADM-RANS</td>
<td>278.0</td>
<td>357.0</td>
</tr>
<tr>
<td>OTF (variable $\nu$)</td>
<td>281.5 (+1.3%)</td>
<td>386.3 (+8.2%)</td>
</tr>
<tr>
<td>OTF ($\nu = 0.1 , \text{m}^2 , \text{s}^{-1}$)</td>
<td>216.7 (-22.0%)</td>
<td>405.3 (+13.5%)</td>
</tr>
<tr>
<td>OTF ($\nu = 1 , \text{m}^2 , \text{s}^{-1}$)</td>
<td>302.1 (+8.7%)</td>
<td>379.9 (+6.4%)</td>
</tr>
<tr>
<td>OTF ($\nu = 3 , \text{m}^2 , \text{s}^{-1}$)</td>
<td>353.8 (+27.3%)</td>
<td>366.8 (+2.7%)</td>
</tr>
</tbody>
</table>

**Table 5.5:** Unsteady flow simulation power integral values. The percentage difference of the OTF data relative to the ADM-RANS data is shown inside the brackets.

However, in the optimised layout the trend is reversed and the highest OTF power integral values are produced with constant $\nu = 0.1 \, \text{m}^2 \, \text{s}^{-1}$ and the OTF constant $\nu = 3 \, \text{m}^2 \, \text{s}^{-1}$ run leads to the lowest power integral value (Table 5.5). The reason for this lies in the fact that in the optimised layout the turbines are predominantly positioned in the bypass flow region of upstream turbines. Once again, increasing $\nu$ leads to shorter wakes, but this also leads to lower bypass flow velocities. Therefore, in the optimised layout, the velocities at the turbine locations decrease as $\nu$ increases. Hence, increasing $\nu$ leads to lower power integral values in OTF, Fig. 5.14b.

Furthermore, in the optimised layout, all OTF simulations overpredict the ADM-RANS data, but the differences are smaller than those observed for the staggered
In fact, the $\nu = 0.1 \text{ m}^2\text{s}^{-1}$ OTF value and $\nu = 3 \text{ m}^2\text{s}^{-1}$ OTF value are only 10% different relative to each other. Given that in this layout the turbines are not positioned directly behind one another, the knock on effect observed in the staggered layout does not apply here and the differences between the various runs are relatively smaller. These differences are primarily due to the 2D representation of truly 3D wake profiles. The match between the ADM-RANS model and the variable eddy viscosity OTF model can be improved by investigating the relationship between $\nu$ and $u_{\text{in}}$ in greater detail. In this study, a simple linear relationship based on the wake profiles at $12D$ has been suggested. Having said that, if the wake profiles at $6D$ are considered, the linear relationship would not change significantly (same gradient but smaller intercept value, Fig. 5.4). However, this should be further examined in order to improve the agreement between the two models. Nonetheless, a good qualitative agreement between the 3D ADM-RANS model and the depth-averaged OTF model has been observed for both the steady and the unsteady flow cases considered in this study. This gives confidence in the value of the optimised layouts obtained via OTF, although higher fidelity 3D models may still be required to accurately estimate the array power outputs and to produce truly optimal array layouts.

Furthermore, although the $\nu = 1 \text{ m}^2\text{s}^{-1}$ and $\nu = 3 \text{ m}^2\text{s}^{-1}$ OTF runs provide a better match than the variable $\nu$ OTF run in the optimised layout, one should bear in mind that this optimised layout was actually achieved via the use of the variable viscosity relationship, Eq. (5.19), in OTF. For example, with $\nu = 3 \text{ m}^2\text{s}^{-1}$ in OTF, there is very little wake interaction in the initial staggered layout and the turbine positions would be hardly altered by OTF. Hence, the optimised layout obtained by using a spurious viscosity value in OTF is unlikely to be very different from the initial staggered layout. Note how the staggered and optimised power integral values for the $\nu = 3 \text{ m}^2\text{s}^{-1}$ OTF runs are only 3.6% different from each other. Therefore, an optimised layout that leads to a 28.4% increase in power integral (according to the ADM-RANS model) would not have been obtained without the use of the variable viscosity OTF model. Hence, it has been demonstrated that employing a suitable calibrated viscosity value consolidates and improves the array optimisation of OTF and helps obtain more realistic optimised layouts.
Figure 5.14: OTF vs ADM-RANS unsteady flow. The solid lines show the ADM-RANS results (left (—) and right (—)) and the dashed lines show the OTF results with a variable viscosity (left ( — ) and right ( — ) ). The dotted lines show OTF results with \( \nu = 3 \text{ m}^2\text{s}^{-1} \) (left ( _ _ _ ) and right ( _ _ _ ) ). The shaded areas represent the range of values obtained if constant eddy viscosity values of \( 0.1 \text{ m}^2\text{s}^{-1} \leq \nu \leq 1 \text{ m}^2\text{s}^{-1} \) are used in OTF.

5.5 Conclusions

In order to assess the suitability of using depth-averaged models to assess the flow past arrays of tidal turbines, a comprehensive comparison between the wake profiles and array power outputs predicted by OTF and the 3D Fluidity ADM-RANS model has been presented. A viscosity sensitivity analysis has been carried out to outline the limitations associated with using a constant viscosity everywhere in the domain in the depth-averaged model. This has highlighted the need to adjust the viscosity value used in the depth-averaged model in-line with the upstream velocity value. Based on this, a simple relationship between viscosity and the upstream velocity value is suggested for the channel flow case.

Furthermore, this relationship was used to compute the appropriate viscosity value to be used in OTF for the flow past an array of 32 tidal turbines in an ideal channel. Both steady and unsteady flow cases have been examined. The OTF package was then used to optimise the positions of the 32 turbines to maximise the power output from the array. The OTF 32 tidal turbine array simulations have been replicated using the Fluidity 3D ADM-RANS model and the total power outputs predicted by the two models have been compared against each other. This allowed for an assessment into the accuracy of the adjoint based optimisation used in OTF for the first time. The results have outlined a good qualitative agreement between the two models highlighting the importance of the correction to the power calculations em-
ployed in OTF as well as the viscosity relationship suggested. This agreement only holds true, if appropriate viscosity values are chosen, highlighting the importance of careful viscosity calibration.

One of the main limitations of the current study is that blockage effects have been neglected in the channel flow cases considered. The inlet velocity is assumed to be independent of the flow blockage inside the channel. Hence, the inlet velocity is unaffected by the turbine layouts. However, in a realistic environment, this is not the case and the upstream velocity is reduced as more and more turbines are added and this will depend on the turbine layout. Therefore, the next step would be to apply a more realistic boundary condition (e.g. fixed head difference) at the inlet. Given that a Dirichlet velocity boundary condition was used in both the 3D ADM-RANS and the depth-averaged OTF simulations, the comparisons presented here are still of great value. Switching to a fixed pressure/head difference boundary condition will alter the power production values and change the optimised layouts, but the qualitative agreement established between 3D ADM-RANS and the depth-averaged OTF models should still hold true, as long as the boundary conditions used in the two models are consistent.

In conclusion, it is not always feasible to incorporate turbulence models into large scale tidal flow simulations. Herein, it has been demonstrated that by using a simple linear relationship for viscosity that scales with upstream velocity, as opposed to using a spuriously high constant value, the disadvantages of not modelling for turbulence directly can be minimised. However, bathymetry effects have been neglected in this study and these will undoubtedly play a crucial role in the ambient turbulence values and consequently the rate of wake re-energisation. A viscosity coefficient that scales with both upstream velocity and bathymetry will help limit the disadvantages of not accounting for turbulence even further. This will lead to significant savings in the computational cost of the large scale simulations.
6.1 Introduction

An ultimate aim in this study was to develop a numerical model which can be used to assess the effects of installing arrays of tidal turbines within realistic tidal sites. One site of particular interest is the Inner Sound of the Pentland Firth. This chapter focuses on the first build-out phase of the MeyGen project in the Inner Sound. This will be the world’s first multi-turbine tidal stream energy project. Phase 1a of the MeyGen project has a capacity of 6 MW and involves installing four 1.5 MW tidal turbines within the Inner Sound. The tidal turbines will be 3 bladed horizontal axis turbines with an 18 m diameter rotor and fully submerged with a minimum of 8 m clearance to the sea surface at the lowest astronomical tide. Following Phase 1a, MeyGen plan to continue to build-out the project to the full 398 MW capacity (MeyGen Ltd., 2015).

In this chapter the basis for embedding the ADM-RANS model, introduced in chapter 2, within tidal flow simulations of the Inner Sound of the Pentland Firth is presented. In order to embed the ADM-RANS model within regional scale simulations, the large scale simulations are carried out in Fluidity using a single layered model for efficiency, where 3D dynamics are unimportant, and in regions where the
turbines are located the number of layers is increased and the ADM-RANS model is used. This allows for an accurate prediction of the power output from the array and it can also provide valuable information on how the global and regional flows are affected by the array of tidal turbines.

6.2 Methodology

In this section the methodology behind the large scale simulations as well as a few modifications made to the ADM-RANS model are presented.

6.2.1 The domain

A map of the MeyGen site within the Inner Sound of the Pentland Firth was presented in the Introduction chapter, Fig. 1.8. However, in order enforce the correct tidal forcing and to be able to capture the effects of the array of tidal turbines on the flow, a much larger domain needs to be considered. Furthermore, the open boundaries need to be far enough from the Inner Sound to minimise boundary effects. Hence, for the simulations presented here, the domain considered encompasses the entire Orkney Islands and a portion of the northern Scottish coastline, Fig. 6.1. The simulation domain is bounded by parallels at N58°12′ and N60°27′. The western side is at the meridian through E3°54′ and the eastern side is composed of constant bearing lines through points (N60°27′, E2°24′) to (N59°, E1°40′30″) and (N59°, E1°40′30″) to (N58°12′, E1°57′36″). This is the same domain considered by Martin-Short et al. (2015) to investigate the potential impacts of large arrays of tidal turbines on flow regime and sediment transport in the Inner Sound. However, therein the depth-averaged shallow water equations were considered and the turbines were represented as individual regions with enhanced bottom drag (Martin-Short et al., 2015). The distance to outer boundaries is sufficient to not influence the undisturbed flow velocity through the site studied, however this may not be sufficient for flow velocity to be unaffected whilst energy is extracted (Adcock et al., 2013).

The surface mesh used in the Inner Sound simulations is shown in Fig. 6.2. It is an unstructured mesh with increased resolution near coastlines and the mesh gradation
to successively higher resolution is also shown. The resolution varies from 10 km at the open boundaries to a minimum edge length of 4 m used in the structured mesh in the region where the turbines will be placed. At run-time Fluidity *extrudes* this 2D mesh down to the bathymetry to create a 3D mesh with a user-specified number of layers. Generally a single layer is used in the simulations, however in the rectangular region where the turbines will be placed, the number of layers is increased to 8. A 2D depth profile of the mesh showing how the number of layers increases at the array site is presented in Fig. 6.3.

For the large scale simulations, eight tidal constituents (M2, S2, K1, O1, N2, K2, P1 and Q1) have been considered here. Similar to Martin-Short et al. (2015), the free surface elevation was reconstructed using the OSU Tidal Software (OTPS, Egbert and Erofeeva (2002)) and applied as a free surface boundary condition along the open boundaries. Furthermore, the bathymetry data used here was obtained from the Edina Digimap service (HydroSpatial One, 2014) with a one arcsec (∼30 m)

![Figure 6.1](image.png)

**Figure 6.1**: Domain considered in the Inner Sound simulations. Taken from Martin-Short et al. (2015). Map of the UK is shown in (A) and the simulation domain is shown in (B). A zoom in of the Pentland Firth and the surrounding islands is shown in (C). The colour relief represents the resampled bathymetry used here.
resolution. In the velocity field a quadratic drag boundary condition, with a non-dimensional drag coefficient of $C_D = 0.0025$, is applied to the bottom surface. This is identical to the bottom boundary condition described in section 3.7.1.

Figure 6.2: The unstructured surface mesh used in the Inner Sound simulations. A higher resolution is used near coastlines and at the array site. The entire domain surface mesh is shown in (a) with a zoom-in of the Pentland Firth shown in (b). A zoom-in of the Inner Sound mesh is shown in (c) demonstrating how the resolution increases at the array site and a zoom-in of the structured array site surface mesh is shown in (d). The locations of the four turbines is also shown in red.

Figure 6.3: A 2D slice across the 3D mesh showing how the number of layers increases at the array site. The turbine (actuator disc) is shown in red.
6.2.2 Absorption term

For the purpose of the simulations presented in this chapter, the ADM source term was implemented as an absorption term in order to aid with the stability of the simulations. In order to do so, the $S_u$ term in Eq. (2.1) is replaced by

\[-\sigma_{ADM}||u||,\]

and therefore the absorption term is defined as

\[\sigma_{ADM} = -\frac{S_u}{||u||}.\] (6.1)

As a result of this, the simulations are more stable and the timestep can be increased to 5 s compared to the 2 s used in the array simulations presented in chapter 5. This has helped speed up the simulations and make the large scale simulations more tractable.

6.2.3 Background viscosity

So far, in the ADM-RANS simulations presented in chapters 3–5, a low background viscosity value of $\nu_{bg} = 1 \times 10^{-6} \text{m}^2\text{s}^{-1}$ has been used. The reason for this lies in the fact that a high value would lead to exaggerated wake re-energisation which would result in shorter wakes as demonstrated in section 5.3. Therefore, a low value was used alongside a RANS turbulence model which varies the eddy viscosity, typically increasing it within regions with a high shear velocity. However, in large scale simulations with complex dynamics throughout the domain a relatively high viscosity value is usually necessary in order to ensure stability. Therefore, for this study it was decided to use a variable $\nu_{bg}$ where a low value is used at the array site, so as not to swamp the viscosities coming out of the RANS model, and a high value is used everywhere else to help with stability. Hence, the background viscosity used gradually increases from a value of $1 \times 10^{-6} \text{m}^2\text{s}^{-1}$ within the array site to a value of $1 \text{m}^2\text{s}^{-1}$ everywhere else over a distance of 0.5 km, Fig. 6.4.
Chapter 6: Inner Sound of the Pentland Firth

6.3 Results

Four turbines with $C_t = 0.85$ are placed southwest of the island of Stroma, resembling the MeyGen Phase 1a site. The turbines' centres are located at a depth of 16 m in a region with an average depth of around 33 m. Mesh optimisation has not been used in these simulations at this initial phase. Hence, due to the structured mesh used at the array site, it was decided to use turbines that are rectangular in shape with a width of 16.5 m, a height of 16 m and a thickness of 4 m. This helps avoid the need for a very fine resolution within the array site in order to resolve the curvature of circular turbines (actuator discs). The actual tidal turbines that will be installed here have a diameter of 18 m and an effective cross-sectional area of around 255 m$^2$. The rectangular turbines used here have a cross-sectional area of 264 m$^2$ which is only 3.5% larger than that of the real turbines. Hence, the limitations associated with assuming non-circular actuator discs are unlikely to significantly affect the results, especially given that the main focus here is on the large scale effects. Moreover, the $k – \omega$ SST RANS model developed and validated in chapters 2–4 is used here to ensure that the effects of ambient turbulence on the wake re-energisation downstream of the turbines are accounted for as well. Furthermore, no spin-up time has been used in these simulations and the flow through the open boundaries is determined by specifying free-surface elevations (i.e. pressure boundary conditions) in-line with the tidal constituents considered, section 6.2.1.

In order to demonstrate the benefits of using RANS models, two different 3D simu-
lations were run. Initially a fixed $\nu = 1 \text{ m}^2 \text{s}^{-1}$ was used everywhere and no RANS model. Thereafter, the simulations were run with the $k-\omega$ SST RANS model and a variable $\nu_{bg}$ described in section 6.2.3 and Fig. 6.4. Each simulation was run in parallel on 240 cores for 48 hours on cx2, a large scale massively parallel processing system housed in Imperial College London’s Data Centre. This yielded over 12 h of data for the fixed viscosity simulation and just under 7 h of data for the $k-\omega$ SST simulation. The extra complexity associated with the $k-\omega$ SST RANS model leads to a higher computational cost and this is evident in the difference in the amount of simulated data retrieved after 48 h of runtime. Having said that, the difference is less than a factor of 2 and therefore the RANS simulations are still computationally feasible.

### 6.3.1 Large-scale flow

Fig. 6.5 shows 2D slices of the 3D domain at turbine hub height (16 m depth) of the flow around Stroma at two different times for both the fixed $\nu$ and the $k-\omega$ SST simulations. Fig. 6.5a – 6.5b show snapshots during the flood flow at $t = 0.83$ h. Note that this is not peak flood flow since the simulations have not been run for long enough to capture a full cycle. Nonetheless, two high velocity regions north and south of Stroma are displayed where the flow is accelerating around the island as it flows west to east. A low velocity region develops on the east side of the island, but no significant wake develops due to the relatively low flow speeds. Fig. 6.5c – 6.5d display snapshots during the ebb flow at $t = 6.67$ h which is very close to the peak ebb flow velocity. Once again, high velocity regions develop north and south of Stroma as the flow interacts with the island as it flows east to west. However, compared to the previous snapshot, a much more significant wake develops to the west of Stroma due to the higher flow velocities present in this case.

As might be expected, the global flows are not significantly affected by the introduction of the $k-\omega$ SST model and the two simulations compute very similar profiles. The variation of $\nu_T$ within the Inner Sound for the $k-\omega$ SST simulation is also shown in Fig. 6.6. The viscosity values are of the same order of magnitude with no significant drop in the values, other than very close to the turbines (which will be discussed in the next section). This explains the close match in the global flows.
Chapter 6: Inner Sound of the Pentland Firth

Figure 6.5: 2D slices of the 3D domain at turbine hub height showing the velocity magnitude variations within the Inner Sound.

(a) $t = 0.83\, h$ ($\nu = 1\, m^2 s^{-1}$)

(b) $t = 0.83\, h$ ($k - \omega$ SST)

(c) $t = 6.67\, h$ ($\nu = 1\, m^2 s^{-1}$)

(d) $t = 6.67\, h$ ($k - \omega$ SST)

Figure 6.6: 2D slices of the 3D domain at turbine hub height for the $k - \omega$ SST simulation showing $\nu_T$ variation within the Inner Sound.

(a) $t = 0.83\, h$

(b) $t = 6.67\, h$
between fixed $\nu$ and the $k - \omega$ SST simulations observed.

The turbulence intensity ($I$) variation within the Inner Sound at hub height is displayed in Fig. 6.7. The turbulence intensity presented in this chapter has been computed based on the local velocity values, as opposed to a fixed upstream value which was the case in the chapters 3 – 5. Hence, the values represent a relative distribution of turbulence within the domain for the particular timestep shown. During the flood flow two regions of high $I$ develop either side of Stroma. The slightly concave curvature on the west of Stroma results in some of the flow getting trapped on the west side and this encourages mixing and results in the high $I$ values observed. On the east, a small wake develops which quickly mixes with the higher velocity flow and this leads to the increase in $I$ east of Stroma. On the other hand, for the ebb flow case, the convex shape east of Stroma means that the majority of the flow passes smoothly around the island without getting trapped and this leads to low $I$ values east of Stroma. Two bands of high $I$ develop north and south of Stroma in the region where the low velocity fluid in the wake mixes in with the higher velocity bypass fluid, Fig. 6.5d.

**Figure 6.7:** 2D slices of the 3D domain at turbine hub height for the $k - \omega$ SST simulation showing $I$ variation within the Inner Sound.
6.3.2 Regional flow

A close-up of the velocity magnitude within the array site at hub height is shown in Fig. 6.8 in order to demonstrate the wake interactions. Here, there is a noticeable difference between the two simulations in the wakes that form downstream of the turbines. In the $k-\omega$ SST simulations the wakes are much longer and more defined, whereas for the fixed $\nu = 1 \text{ m}^2\text{s}^{-1}$ case the wakes are more dissipative. The variation of $\nu_T$ within the array site for the $k-\omega$ SST simulation is also shown in Fig. 6.9. The effect of viscosity on the rate of wake re-energisation was already discussed in detail in section 5.3 where it was demonstrated that a higher viscosity coefficient leads to faster wake re-energisation. In the $k-\omega$ SST simulation, within the array site there is a significant variation in the $\nu_T$ with the values dropping almost two orders of magnitude in the wakes downstream of the turbines. This leads to slower wake re-energisation which explains the less dissipative and more defined wake profiles observed in the $k-\omega$ SST results.

Variation of $I$ within the array site at hub height is shown in Fig. 6.10. As expected, there is an increase in $I$ downstream of the turbines as mixing occurs and the wakes re-energise. This is similar to the wake profiles observed in chapters 3–4. Interestingly, the ambient $I$ is higher for the flood flow compared to the ebb flow despite the lower velocities. The reason for this lies in the fact that the $I$ values shown are computed relative to the local velocity values. The actual turbulence kinetic energy is higher in the ebb flow and this can be seen by comparing the $k$ values at each timestep, Fig. 6.11. Furthermore, the ambient $I$ in both timesteps shown is only around 6% at $t = 0.83 \text{ h}$ and 4% at $t = 6.67 \text{ h}$. Both of these values are lower than the actual $I$ within the Inner Sound which is expected to be more turbulent (Goddijn-Murphy et al., 2013).

Contrary to the previous chapters where $k$ and $\omega$ were specified at the inlet as Dirichlet boundary conditions, in the SST simulation presented here the $k$ and $\omega$ boundary conditions were simply set to zero flux. One of the reasons for this is that $k$ and $\omega$ values at the open boundaries are unknown. More importantly, the open boundary are chosen to be far away enough not to affect the flow in the Inner Sound and therefore any value specified would not significantly alter the values within the Inner Sound. Instead, the ambient $k$ and $\omega$ values, which impact the ambient $I$,
6.3: Results

(a) $t = 0.83 \text{ h} \ (\nu = 1 \text{ m}^2 \text{s}^{-1})$

(b) $t = 0.83 \text{ h} \ (k - \omega \text{ SST})$

(c) $t = 6.67 \text{ h} \ (\nu = 1 \text{ m}^2 \text{s}^{-1})$

(d) $t = 6.67 \text{ h} \ (k - \omega \text{ SST})$

Figure 6.8: 2D slices of the 3D domain at turbine hub height showing the velocity magnitude variations within the array site.

(a) $t = 0.83 \text{ h}$

(b) $t = 6.67 \text{ h}$

Figure 6.9: 2D slices of the 3D domain at turbine hub height for the $k - \omega \text{ SST}$ simulation showing the $\nu_T$ variations within the array site.

are determined by the level of turbulence generation within the domain as the flow meanders around the Orkney islands. However, it appears that the simulation has
underpredicted the level of turbulence generation or possibly overpredicted the rate of decay of turbulence within the domain and this has led to the reduced ambient $I$ values observed. One of the main issues with the current Fluidity model is the absence of an appropriate bottom boundary condition that can accurately model the generation of turbulence. The limitations associated with the drag boundary condition used here have been previously discussed in section 3.7.1. Hence, once this limitation is addressed via the use of suitable wall functions, the ambient $I$ values of the simulations will provide a closer match to the real values recorded experimentally.
6.3: Results

6.3.3 Depth profile

As mentioned earlier, the main advantage of the 3D simulations is that the variation with depth is modelled. In order to demonstrate this, velocity and $I$ depth profiles of the flow past turbine 2 at $t = 6.67\,\text{h}$ for the $k - \omega$ SST simulation is shown in Fig. 6.12. Note that since the wake curves around Stroma, the profile shown is not exactly along the centreline of the wake. In the velocity profile, Fig. 6.12a, the upstream flow is almost uniform, but there is a slight reduction in velocity near the bottom due to the drag boundary condition applied. Having said that, the velocity does not drop off to zero since the boundary layer is not modelled here. Notice that the flow accelerates underneath and above the turbine. This is more pronounced above the turbine whereas the flow is not able to accelerate as much underneath the turbine due to bottom drag. Consequently, an anti-symmetric wake forms downstream of the turbine which re-energises as the bypass flow mixes in with the wake. Furthermore, the wake appears to be longer than those observed in chapter 4 and does not recover as quickly.

![Velocity magnitude and turbulence intensity profiles](image)

**Figure 6.12:** Depth profiles showing the flow past turbine 2 at $t = 6.67\,\text{h}$ for the $k - \omega$ SST simulation.

Moreover, the upstream $I$ profile is not exactly uniform either, Fig. 6.12b. The reduction in velocity near the bottom leads to horizontal shear which in turn results in increased $I$ near the bottom. As the flow passes the turbine, a high $I$ develops in the wake which slowly dissipates as the wake re-energises. Once again, the ambient $I$ values observed are lower than those present in real tidal flows. Higher ambient $I$, which is expected in the Inner Sound, would lead to quicker wake re-energisation and this should result in shorter wakes compared to those shown in Fig. 6.8.
6.3.4 Power

The power outputs of the four turbines throughout the simulation are shown for both the fixed $\nu$ and the $k-\omega$ SST runs in Fig. 6.13. Note that a constant $C_t$ has been used here and therefore the power has not been capped. Hence, the peak power values shown exceed the 1.5 MW maximum. This will not be the case in the real world as the turbine are designed not to generate power above the rated value. The combined power output of all four turbines is also shown in Fig. 6.14. The velocity values are not shown, but since power scales as velocity cubed and no power capping is applied, the velocity history plot will be very similar to the power history plots shown, Fig. 6.13–6.14.

Generally, the two simulations follow the same pattern with the fixed $\nu$ simulation consistently predicting higher values. This is due to the faster wake re-energisation predicted in the fixed $\nu$ which leads to higher velocity values at the turbines. This in turn leads to greater power values. Furthermore, the power output from turbine 1 is significantly higher than that of the other three turbines, especially during peak ebb flow (around $t = 6$ h). This suggests higher velocity values during ebb flow at the location of turbine 1. Close examination of Fig.6.8c–6.8d reveals that the flow curves slightly around Stroma as it flows east to west. As a result of this turbine 1 ends up in the bypass flow region of turbine 3 and this positive feedback results in higher velocity values at turbine 1 during ebb flow and this leads to greater power values.

6.4 Conclusions

In this chapter a methodology for embedding the 3D ADM-RANS model within large scale simulations with real tidal forcing has been outlined. In order to do so efficiently a single layered mesh has been used with the number of layers increasing within the array site, where 3D dynamics are important. The benefits of such an approach have been outlined by simulating Phase 1a of the MeyGen project which involves installing four turbines southwest of the island of Stroma. This is the first time large scale simulations of real tidal flows past an array of tidal turbines within the Inner Sound has been carried out in 3D, to the best knowledge of the author.
The 3D simulations allowed the flow passing underneath and over the turbines to be modelled and this has led to predictions of the velocity and free surface variations within the Inner Sound. Moreover, the $k - \omega$ SST RANS model was used to account for the effect of turbulence on the flow, thus improving the modelling of the wake recovery, which is known to be sensitive to ambient turbulence characteristics. By comparing the results from the RANS model to a fixed $\nu = 1 \text{ m}^2\text{s}^{-1}$ simulation, the limitations of using a spurious viscosity value everywhere was outlined. Most notably, overprediction of the rate of wake re-energisation which leads to exaggerated
Chapter 6: Inner Sound of the Pentland Firth
	power production values was avoided by using the $k - \omega$ SST RANS model.

The simulations have not been run for long enough to capture a full cycle and the results presented here should be regarded as a feasibility study only. In order to give confidence in the velocity and power production values presented in this chapter, a mesh sensitivity study needs to be carried out, at least for the mesh used within the array site. Moreover, mesh optimisation can be enabled to help alleviate mesh sensitivity concerns. This might increase the computational cost of the simulations, but a drastic increase in computational cost can be avoided if the mesh optimisation is limited to a small area (e.g. only over the array site). This would also enable the use of circular turbines (actuator discs) instead of the rectangular ones currently used. Furthermore, one of the main limitations of the current model is the under-predicted ambient $I$ values which occur due to the absence of appropriate bottom boundary conditions. However, the introduction of wall functions for the velocity, $k$ and $\omega$ fields should help address this problem and increase the ambient $I$ values bringing them closer to the values expected within the Inner Sound.
This thesis focuses on the extraction of tidal stream energy from coastal waters via horizontal axis tidal turbines which are currently one of the favoured approaches to efficiently harness the vast and reliably predictable tidal resource. The deployment of tidal turbines is a complex and expensive operation and this makes the task of locating the optimal position for such turbines even more important. The main aims of the research reported were the development and evaluation of a multi-scale model capable of simulating the flow through such arrays and the consequent power performance of alternative turbine configurations.

The general contributions and the main findings of this research are summarised below and are expanded upon in the subsequent text.

- the effects of ambient turbulence intensity on the wake structure cannot be ignored, even at the array scale.

- the short-circuiting of the turbulence cascade due to the presence of the turbine needs to be accounted for.

- the viscosity value chosen in depth-averaged models needs to be carefully chosen to reflect the ambient turbulence conditions.
via the combination of an appropriate RANS model and mesh optimisation techniques the flow past arrays of tidal turbines can be accurately and efficiently simulated.

- developed a numerical tool that can bridge the gap between 3D turbine scale models and 2D depth-averaged regional scale models.

The numerical simulation of arrays of tidal turbines is a highly multi-scale problem. The scales of interest vary from a few metres at the turbine scale, where the flow past a single turbine is studied, to a few hundred kilometres at the regional scale, where tidal flows within realistic domains are considered. Herein, the highly parallelisable multi-scale Fluidity solver is used to develop an ADM-RANS model. The model uses actuator disc momentum (ADM) theory and Reynolds-averaged Navier-Stokes (RANS) with, for the first time, dynamically adaptive mesh optimisation techniques. Hence, the mesh can be refined dynamically in time and only in the locations required, thus making optimal use of limited computational resources. Furthermore, in a realistic environment the ambient turbulence intensity has a significant effect on the structure of the turbine wake and its recovery. Therefore, both $k – \omega$ and $k – \omega$ SST RANS models have been developed within the Fluidity framework in order to account for the effects of turbulent mixing.

The ADM implementation is initially verified against a semi-analytical solution in section 2.3. The Fluidity model is also compared against a similar ADM model developed in OpenFOAM in order to demonstrate the advantages of the finite element discretisation employed in the Fluidity ADM-RANS model. It is demonstrated that the Fluidity model performs better at capturing the sharp velocity variation across the actuator disc, whereas the OpenFOAM model exhibits some fluctuations at a consistent mesh resolution.

An experimental study of the flow past an actuator disc rotor simulator is carried out in chapter 3 in order to highlight the shortcomings of the assumptions used in the development of the ADM-RANS model. It is demonstrated that ADM theory presents a robust tool to model the flow past porous discs. The shear layer between the wake and the bypass flow where the higher momentum fluid mixes in with the almost stagnant fluid, visible in the experimental results, is correctly captured via the ADM-RANS model. This is crucial since this mixing is the main
mechanism with which the wake is re-energised. Overall, the simulations provide a good agreement in the far wake and given that this study focuses on modelling arrays of turbines, the far wake profiles are of primary importance here. However, the quadratic drag boundary condition used in the ADM-RANS model is shown to lead to minimal bottom turbulence generation. Hence, in future work wall functions need to be introduced into the ADM-RANS model in order to improve the accuracy of the numerical results.

The suitability of the RANS turbulence models developed for this study is further explored in chapter 4 by recreating two different experimental flume tests where scaled turbines are used. The ADM-RANS model does not take into account the swirl induced on the flow by the rotating blades used in these experiments, but it is still able to produce closely matching results by capturing the important turbulence characteristics in the far wake. The turbine scale simulations outlined the importance of accounting for the short circuiting of the turbulence cascade due to the presence of the turbine when using RANS models. Furthermore, mesh optimisation techniques are successfully employed to obtain an accurate representation of the turbine wakes whilst maintaining a relatively low computational cost.

The validated 3D ADM-RANS model is then used to examine the suitability of using depth-averaged models to assess the flow past arrays of tidal turbines. An eddy viscosity sensitivity analysis is carried out and the wake profiles predicted by the depth-averaged OTF package and the 3D Fluidity ADM-RANS model are compared against each other in order to outline the limitations associated with using a constant viscosity everywhere in the domain in the depth-averaged model. This highlighted the need to adjust the viscosity value used in the depth-averaged model in-line with the upstream velocity value. Hence, a simple relationship between viscosity and the upstream velocity value is suggested. This relationship is then used to compute the appropriate viscosity value to be used in OTF for the flow past an array of 32 tidal turbines in an ideal channel where both steady and unsteady flow cases are examined. The OTF package is used to optimise the positions of the 32 turbines to maximise the power output from the array, and the depth-averaged simulations are then replicated using the 3D Fluidity ADM-RANS model. This allowed for an assessment into the accuracy of the adjoint based optimisation used in OTF for the first time. The results outlined a good qualitative agreement be-
Chapter 7: Conclusions

tween the two models, highlighting the importance of careful viscosity calibration in depth-averaged models at the array scale. However, bathymetry effects have been neglected and in order to come up with a comprehensive viscosity definition or turbulence modelling approach to be used in large scale simulations, the analysis should be extended to investigate the effects of bathymetry and proximity of coastlines on eddy viscosity as well.

An ultimate aim was to use the numerical model in realistic large scale simulations. Hence, a methodology for embedding the 3D ADM-RANS model within large scale simulations with real tidal forcing is presented in chapter 5. In order to do so efficiently a single layered mesh is used with a larger number of layers (8) used over the array site. The benefits of such an approach is explored by simulating Phase 1a of the MeyGen project which involves installing four turbines southwest of the island of Stroma. The 3D simulations allowed the bypass flow passing underneath and over the turbines to be modelled. Moreover, the $k - \omega$ SST RANS model is used to account for the effect of turbulence on the flow, thus improving the modelling of the wake recovery, which is known to be sensitive to ambient turbulence characteristics. However, the regional scale simulations have only covered one ebb flow and a complete tidal cycle was not simulated. Therefore, the analysis should be extended in the future to cover a longer tidal period. Moreover, in order to give confidence in the velocity and power production values predicted by the ADM-RANS model, a mesh sensitivity study needs to be carried out. Thereafter, the analysis can be extended to model large arrays of tidal turbines in the Inner Sound in order to investigate their effect on the regional and global flows in the Pentland Firth.

This work can aid future investigations into the optimisation of tidal stream power and help increase the MRE sector’s contribution to the UK’s energy production. The ADM-RANS model developed herein provides a multi-scale numerical tool that can successfully bridge the gap between high-fidelity 3D turbine scale models and 2D depth-averaged regional scale models. By focusing on the major factors which affect the far wake profile and by utilising dynamic mesh optimisation techniques, the model ensures that the wake interactions are accurately captured without the need for excessive computational power. Hence, the 3D ADM-RANS model provides a computational tool that is well suited to the simulation of large-scale tidal turbine arrays, including at locations in UK waters. Furthermore, the model can potentially
be combined with the adjoint-based optimisation techniques used in OTF to develop a 3D array optimisation solver which takes into account the effects of turbulent mixing as well as real tidal forcing and bathymetry data.
Chapter EIGHT

Future work

In order to ensure further contributions to knowledge can be made by advancing this work, the limitations of the current study need to be identified. Here is a list of how the work presented in this thesis can be taken forward:

- appropriate wall boundary conditions need to be implemented in Fluidity to ensure correct bottom turbulence generation.
- the introduction of thrust curves will enable more realistic power predictions, especially at the regional scale.
- the validity of the regional scale simulations needs to be assessed in detail.
- regional scale simulations need to be extended to cover a full tidal cycle.
- mesh optimisation techniques applied locally can help reduce the cost of the regional scale simulations.

There are two aspects of future work which will be addressed in turn. Firstly, the main findings of the experiments described in chapter 3 are summarised and used to plan future experiments on a larger scale. Secondly, the limitations of the ADM-RANS model are summarised and possible improvements and extensions are suggested.
Chapter 8: Future work

8.1 Experimental

The majority of experimental studies carried out have focused on a single turbine or a small array of turbines (Myers and Bahaj, 2010; Blackmore et al., 2014; Mycek et al., 2014a,b; Stallard et al., 2013). This is mainly due to the limitations associated with the facilities available. Expanding such methodologies to analyse large arrays of tidal turbines and their interactions can be extremely valuable as it would help increase our understanding of the physical phenomena associated with these interactions. It can also be used as a valuable tool in validating results from numerical simulations.

There are currently plans underway to build a new wide flow tank in the hydrodynamics laboratory of the Civil and Environmental Engineering Department at Imperial College London. This tank will be 26.9 m long and 6.2 m wide with a maximum water depth of 1.2 m. The wide flow tank can be used as a current flume and will also house wavemakers. It can therefore recreate tidal waves as well as tidal currents which makes it an ideal platform to experimentally analyse tidal flows. Furthermore, the large width of the tank allows for a number of scaled turbines or porous discs to be positioned side by side which allows for the analysis of large arrays.

The experimental procedure described in this chapter 3 was designed in order to simulate and capture the main flow features present in the wake of a tidal turbine. The limitations have already been discussed in section 3.8. However, despite the shortcomings of the current methodology, it is still able to correctly capture the main flow features of the wake. Therefore, it is suitable to extend this to experimentally record the wake profile behind a large array of tidal turbines where each turbine is represented via a 0.15 m porous disc.

Unfortunately, due to building works in the hydrodynamics laboratory linked to the construction of the new wide tank, time on the 4ft flume was limited. Hence, there was not enough time to record the wake profile behind the 50% porous disc as well. Given that this disc has a lower $C_t$, which is closer to the range of values quoted for tidal turbines, it will be helpful to experimentally capture this wake profile and compare it to the one already obtained. This will also help decide on which disc should be used in future experiments where the flow past multiple turbines will be
Once a suitable porosity has been selected and finalised, experimental assessment of the flow past tidal turbine arrays can begin. Fig. 8.1 illustrates a 32 turbine staggered layout where the turbines have been positioned within a 6 m by 3 m rectangular domain at the centre of the new wide flume. Capturing the wake profile in this case will have to be done in stages. In order to do so, initially the first row of turbines will be placed inside the flume and the ADV will be used to measure the wake profiles downstream. The wakes up until the location of the next row of turbines will be recorded. Then, the second row will be placed inside the flume as well and the wake profile downstream of this second row up until the location where the third row is to be placed will be recorded. This is continued until all nine rows are in place and the complete wake profile is obtained.

Figure 8.1: An illustration of a 32 turbine array layout inside the new wide flume currently under construction.

One of the issues with this procedure is that it assumes the discs only influence the flow downstream and do not alter the flow behaviour upstream. This is not necessarily true, but it can be argued that changes upstream are negligible in comparison to the velocity deficits that occur downstream of the discs. Apart from the velocity profiles, the loading on each disc will have to be recorded. The inflow velocity for each disc will be different and therefore the loadings will not be the same, hence it will be interesting to see how the loading varies within the array. This will also provide information regarding the possible power output per turbine within the array as it would indicate regions within the array where there is a greater momentum flux.
Chapter 8: Future work

Note that the experiment described above is a 1:125 scaled down version of the 32 turbine channel flow simulations considered in section 5.4. Hence, the experimental results obtained can be compared against the results from the 3D ADM-RANS model and the depth-averaged OTF simulations. A possible extension would be to then re-arrange the porous discs into the optimal layouts observed in section 5.4 and consequently compare the thrust and power values from the numerical results against experimentally measured values. This will allow for a thorough experimental assessment of both the ADM-RANS model and the adjoint-based optimisation used in OTF.

Figure 8.2: The depth profile inside the flume can be altered by covering the flume bed with marbles.

Figure 8.3: Boundary layer profile: Normal flume vs flume covered with marbles measured at the centre of the flume.

Furthermore, the importance of ambient turbulence intensity on the wake has al-
ready been stressed. The 4ft flume exhibits a lower $I_{\text{exp}}$ than the real environment. However, this can be increased for future experiments. One method will be to remove the honeycomb meshes placed at the inlet which help streamline the flow as it enters the flume. Alternatively, the bottom roughness can be increased by covering the flume bed with marbles, Fig. 8.2. The streamwise velocity and $I_{\text{exp}}$ variations with depth were measured at the centre of the flume covered with marbles and the results are presented in Fig. 8.3 alongside those obtained for the normal flume (section 3.4, Fig. 3.3). The introduction of the marbles has altered the depth profile and resulted in a thicker boundary layer, but the ambient turbulence intensity is not significantly affected. Hence, it is possible to experimentally examine the effect of varying the ambient turbulence intensity on the wake interactions downstream of an array of porous disc rotor simulators by varying the bottom roughness of the flume bed. However, in order to significantly increase the ambient turbulence larger marbles and a rougher flume bed will be required.

8.2 Numerical

One of the main issues that was consistently observed throughout this thesis was the lack of turbulence generation from flume/sea bed. Hence, in order to improve the ambient turbulence computation, wall functions for the $k$ and $\omega$ fields, similar to those available in commercial CFD packages (e.g. Ansys CFX), need to be introduced to improve the bottom boundary representation of the current Fluidity ADM-RANS model. This will help improve the accuracy of the turbulence modelling of the current model.

Furthermore, the versatility of the ADM-RANS model needs to be tested further via a comparison against established higher fidelity numerical models. For example, a Fluidity model capable of mesh optimisation based on ADM that uses LES for turbulence modelling has been developed by Creech et al. (2012) to assess the performance of wind turbines, and Afgan et al. (2013) use both RANS and LES with a 3D moving mesh model of a tidal turbine to investigate wake profiles.

Another worthy addition would be the use of thrust curves, where $C_t$ is expressed as a function of the unperturbed upstream velocity, instead of the constant $C_t$ that
is currently used in the ADM-RANS model. This represents the performance of a real device more accurately as it enables the model to take into account the cut-in speed below which the turbine does not operate and the rated speed above which $C_t$ decreases to maintain a constant power yield. This would be a worthy addition to the model when simulating realistic scenarios.

Once the bottom boundary limitations are addressed and thrust curves are introduced, the ADM-RANS model will be ideally suited to tackle regional scale simulations of arrays of tidal turbines within realistic tidal sites around the UK. Moreover, the mesh optimisation techniques described in this work can be used to ensure an accurate representation of the dynamic wake interactions. This might increase the computational cost of the regional scale simulations, but a drastic increase in computational cost can be avoided by limiting the mesh optimisation to a relatively small area (e.g. only over the array site).


Bibliography


