Faculty of Science and Engineering

School of Engineering, Computing and Mathematics

2019-08-28

Developing a coupled turbine thrust methodology for floating tidal stream concepts: Verification under prescribed motion

Brown, Scott Andrew

http://hdl.handle.net/10026.1/14879

10.1016/j.renene.2019.08.119 Renewable Energy Elsevier

All content in PEARL is protected by copyright law. Author manuscripts are made available in accordance with publisher policies. Please cite only the published version using the details provided on the item record or document. In the absence of an open licence (e.g. Creative Commons), permissions for further reuse of content should be sought from the publisher or author.

Developing a coupled turbine thrust methodology for floating tidal stream concepts: Verification under prescribed motion

S.A. Brown^{a,*}, E.J. Ransley^a, D.M. Greaves^a

^aSchool of Engineering, Computing and Mathematics, University of Plymouth, Plymouth, United Kingdom, PL48AA

Abstract

Floating systems offer an opportunity to expand tidal energy resource through an increase in viable sites and greater flow speeds near the free surface. However, the close proximity of the free surface provides uncertainty regarding power delivery and survivability due to the presence of waves, which could be addressed through a numerical model that is capable of considering all components of a floating tidal system simultaneously. This paper presents the first step in the development of such a tool: using the open-source CFD libraries of OpenFOAM as a basis, a computationally efficient HATT model has been developed for generalised incident flow conditions using actuator theory. A thorough evaluation of the model's sensitivity to key considerations in the simulation of entire floating tidal systems, such as flow speed and mesh alignment, showed that the model is robust, ensuring that it is suitable for future extension to wave-driven environments and integration into a framework for such systems.

Keywords:

HATT model, CFD, OpenFOAM, Actuator theory, Marine renewable energy, Velocity deficit

1. Introduction

Development of the Offshore Renewable Energy (ORE) sector is of high national importance for the UK and tidal stream represents a renewable energy source with a number of desirable characteristics: it is more predictable than other sources (such as wind and wave energy) providing simplified power grid management; the resource tends to be concentrated by topography resulting in desirable sites with high energy densities close to land masses (and to end users), reducing costs in terms of installation and maintenance as well as cabling, and; the majority of present device concepts, particularly Horizontal Axis Tidal

 $^{^{*}}$ Corresponding author

Email address: scott.brown@plymouth.ac.uk (S.A. Brown)

¹⁰ Turbines (HATTs), benefit from technology that has been developed through ¹¹ existing industries, such as hydro and wind power, accelerating the maturity of ¹² the tidal stream industry.

However, the majority of the proposed tidal stream devices (particularly 13 those with the highest level of technology readiness) tend to be seabed-mounted 14 or gravity-based devices. Use of these concepts limits the number of viable sites 15 as the water depth has to fall within a narrow range, due to constraints on the in-16 stallation and the required clearance above the turbine blades. The bathymetry 17 also has to be favourable, i.e. relatively flat and horizontal. Furthermore, due to 18 boundary layer effects, the flow speed at depth tends to be lower and so seabed 19 mounted devices do not exploit the full tidal stream resource at deeper sites, 20 providing a further constraint on water depth. Finally, seabed-mounted devices 21 tend to suffer from time-consuming and difficult installation, maintenance and 22 recovery procedures, greatly increasing the overall cost of the projects. 23

Consequently, a number of floating tidal stream concepts have been pro-24 posed. These concepts have a number of distinct advantages over seabed-25 mounted devices. Floating devices are not limited by water depth, bathymetry 26 or the presence of mobile sediments resulting in a greater number of potential 27 sites and hence a higher potential extractable resource. In addition to this, 28 floating tidal stream concepts tend to be easier and quicker to install, maintain 29 and recover since the majority of them can be towed to site using basic tug 30 boats, reducing the need for expensive specialist vessels. Floating devices also 31 benefit from positioning the turbine towards the top of the water column where 32 the flow velocity is maximum, again increasing the available resource compared 33 with seabed-mounted devices at certain sites. 34

However, despite the advantages of floating tidal stream concepts, it should 35 be noted that sites ideally suited for bottom mounted turbines tend to be shal-36 lower, and hence generally experience faster flow speeds than deeper sites. Fur-37 thermore, the flow observed by bottom-mounted and floating designs would be 38 similar at these shallower sites since the location of the turbine will be closer to 39 mid-water in both cases, limiting the advantage of a floating approach. Float-40 ing devices also require additional considerations to be made, regarding their 41 location at the free-surface: firstly, these devices have an additional impact on 42 other stakeholders at the site, e.g. obstruction of navigation and visibility is-43 sues, and; secondly, these devices are exposed to free-surface effects and waves 44 leading to concerns over both the power delivery and the survivability of the de-45 vices. With so few deployments and limited operating hours to date, the effect 46 of proximity to the free-surface and wave-induced motion/loading on these de-47 vices is not presently understood and crucial, under-pinning research is required 48 before these devices will become commercially viable. 49

As with other emerging ORE industries, modelling (both physical and numerical) has now become an essential part of the development process. Numerical modelling, in particular, is being relied upon more and more to overcome the costs and scaling issues associated with physical modelling as well as to provide the high resolution measurements and the quantitative descriptions required for engineering design.

The modelling of floating tidal stream concepts, however, is incredibly com-56 plex, combining: complicated hydrodynamics, such as the interaction between 57 waves and currents; floating structure; mooring system, and; (possibly multiple) 58 submerged turbines. Existing numerical codes are rarely capable of including 59 all of these elements and, for those that are, the behaviour tends to be linearised 60 and each of the elements treated separately, i.e. a 'decoupled' model. This cre-61 ates considerable uncertainty in the power delivery and survivability predicted 62 by these models. A model which fully resolves the contribution of all elements 63 simultaneously as well as the fully nonlinear hydrodynamics and floating-body 64 motion is therefore desirable when assessing the behaviour of these devices, the 65 loads (in the mooring lines for example) and the power output from the tur-66 bine. Unfortunately, such a model, if available, would likely suffer from excessive 67 CPU requirements making the use of such a tool prohibitive in routine design 68 processes. 69

Therefore, this paper details the first step in an incremental development 70 of an efficient numerical tool that is capable of predicting the fully nonlinear, 71 coupled behaviour of floating tidal stream systems. The article concentrates 72 on the methodology used to generate a computationally efficient HATT model 73 that predicts accurately the coupled forces on the turbine, and the fluid, while 74 remaining numerically stable under arbitrary motion. Using the open-source 75 Computational Fluid Dynamics (CFD) libraries of OpenFOAM as a basis, the 76 HATT model has been developed for generalised incident flow conditions using 77 actuator theory. It should be noted that actuator approaches have been im-78 plemented in OpenFOAM in previous work, but they have largely focused on 79 validation of fixed wind turbine wake predictions [25, 26, 33, 44], and the impli-80 cations for wind farm layout [32, 49]. Although these previous methods for fixed 81 turbines provide a basis for floating applications, they are not directly applicable 82 since they often require a very specific mesh layout to maximise alignment with 83 the turbine, which could not be achieved if the turbine position is constantly 84 updating. Therefore, the approach presented here incorporates the effects of the 85 turbine model on the fluid dynamics in the fully nonlinear Reynolds-Averaged 86 Navier-Stokes (RANS) solver via a 'body-force', momentum-sink-type method-87 ology which allows the turbine position to move independently of the mesh. 88 This results in a strongly coupled model that is rigorously characterised, using 89 steady-state simulations, and demonstrated to be robust in a series of test cases 90 in which the turbine has prescribed motion. 91

92 2. Background

High-fidelity numerical methods, such as CFD, have been used extensively in mature industries, like wind energy, to assess the behaviour and performance of horizontal axis turbines. The development of tidal stream turbines has benefited greatly from the knowledge gained in the wind industry, however; it is important to recognise that tidal turbines can be subject to free-surface effects (such as ventilation), possible cavitation and bi-directional flow and that the established methods must be adapted to provide an accurate prediction of tidal stream turbine behaviour [35]. Moreover, existing methods rarely take into account the
arbitrary motion of a freely moving turbine which, in the case of a floating tidal
stream device, could result in a turbine velocity comparable to the free-stream
velocity of the fluid. The following is a brief review of the most commonly used
methods for turbine modelling in CFD.

Arguably, the most realistic methods are 'bladed-resolved' techniques, in 105 which the turbine is directly meshed into the computational domain, allow-106 ing the flow to be resolved as it passes the turbine blades [15, 19, 21, 43, 50]. 107 Computationally, these methods are extremely expensive; the spatial resolution 108 must be fine enough in the vicinity of the turbine to accurately model both the 109 complex geometry of the turbine and the small-scale flow structures around the 110 blades; and the time step must be shorter than the temporal scales of these 111 small-scale structure, in order to resolve them accurately. Furthermore, the 112 mesh needs to be updated at every time step to accommodate the rotation of 113 the turbine, which often involves complex remeshing techniques (e.g. arbitrary 114 mesh interface (AMI) [46]), or additional interpolation overheads (e.g. overset 115 grid [21, 50]), further reducing the computational efficiency. Thus, although 116 potentially very accurate, the computational cost of blade-resolved methods is 117 often considered to be prohibitive in routine design processes and so cheaper al-118 ternatives have been developed, aiming to represent the required characteristics 119 of the turbine without the need to resolve the flow around the turbine blades. 120

Actuator methods are a common approach when representing a horizontal 121 axis turbine in a fluid flow. The simplest cases are actuator disc models which, 122 based on momentum theory, apply a 'resistance' to the incoming flow over the 123 swept area of the turbine, similar to that of a porous disc (which is often used in 124 physical laboratory experiments as a simple representation of a turbine [3, 22, 125 31). The applied resistance typically takes the form of a momentum sink, the 126 magnitude of which is based on the relationship between the free-stream velocity 127 and the thrust on (and power generated by) the turbine [6]. In these methods 128 a coarser, static mesh can be used, greatly reducing the computational costs 129 relative to blade-resolved methods. A number of authors have utilised actuator 130 disc models, in a wide range of numerical models including CFD simulations of 131 both wind [2, 7, 36, 45] and marine current [1, 4, 5, 13] turbine applications. 132 However, it has often been found that an actuator disc approach suffers from 133 the absence of rotational effects (particularly when the focus is on an accurate 134 prediction of the turbine wake [13, 12, 38]). Since the area of the actuator 135 disc is fixed, vorticity is shed into the wake as a continuous sheet from the 136 edges of the disc instead of from the tips of the blades [38]. To increase the 137 accuracy in unsteady flows and capture rotational effects, extensions to the 138 actuator disc methodology have been developed. These include: actuator line 139 [8, 16, 29, 42, 41] and actuator surface [20, 40, 47] methods in which the applied 140 momentum sink is distributed into finite lines or surfaces to represent the blades 141 of the turbine. Furthermore, in these methods the momentum sink is considered 142 to be transient with the position of the blades being updated based on the flow 143 speed and the characteristics of the turbine. In these methods, the torque on 144 the generator can then be calculated from the angular velocity of the rotation. 145

In addition, by discretising the blades in this way, distinct tip vortices can be
calculated (rather than the continuous vorticity sheet arising from an actuator
disc model) giving an improved representation of the turbine wake [38].

Blade element momentum theory (BEMT) [4, 12, 24, 27] is another extension 149 of the actuator disc model, combining blade-element and actuator methodologies 150 to calculate the lift and drag forces on each section of the discretised turbine 151 blades [6]. With the inclusion of a 'tip loss correction factor' [27, 30, 39], which 152 accounts for vortex effects at the blade tips, BEMT has been shown to have 153 good agreement with physical measurements and blade-resolved CFD models 154 [23, 27, 24]. Masters et al. [27] also suggests that further improvements can be 155 made by including a 'hub loss correction factor' [30] which, in a similar way to 156 the tip loss correction, accounts for vortex effects caused by the presence of the 157 rotor hub. 158

159 3. Methodology

A new library, allowing for the representation of tidal turbines, has been 160 designed and implemented in OpenFOAM (v. 4.1 [48]), an open source tool-161 box aimed at solving continuum mechanics problems (including CFD). The 162 software is written in C++ and is based around the Object Orientated Pro-163 gramming (OOP) paradigm, offering a large collection of solvers and shared 164 libraries. Consequently, the new turbine library is easily coupled with many 165 of the existing solvers. However, in this study the focus is on three solvers 166 of increasing complexity, that solve the incompressible RANS equations using 167 the Finite Volume Method (FVM): simpleFOAM, for steady-state simulations; 168 pisoFOAM, for transient single fluid cases; and interFOAM for simulating free 169 surface flows using a two-phase Volume Of Fluid (VOF) approach [37]. 170

Since it is computationally expensive to resolve the flow structure around 171 the turbine, a simpler, more efficient approach for modelling the turbine is 172 adopted (compared with a blade-resolved method). Furthermore, since the over-173 arching aim of this work is to develop a tool for modelling complete floating tidal 174 stream devices, the focus here is on facilitating the key aspects required for a 175 coupled system, i.e. accurate prediction of the coupled forces and numerical 176 stability with arbitrary motion of the turbine, rather than on developing a new 177 turbine model. Therefore, in this study, an actuator disc model has been used 178 to demonstrate the methodology (it is, however, worth noting that, due to the 179 object oriented nature of the code developed here, it is relatively straightforward 180 to include more sophisticated turbine models, such as a BEMT approach, in the 181 future). 182

To allow for arbitrary movement of the turbine through the computational domain, the turbine model here is based upon a 'weighted body force implementation' which, at each time step, identifies and applies weights to a finite 'region' of the computational domain (representing the turbine). This requires no constraints on the local mesh structure (a requirement for the complete, coupled device), contrary to common methods used in static cases which often require the mesh to be highly contrived in the disc region [7, 12, 24]. These weights are then used to determine the local velocity at the turbine position as
well as to add an additional, equal and opposite force (based on the thrust on
the turbine) to the momentum equations, ensuring that the model is two-way
coupled.

¹⁹⁴ 3.1. Actuator Disc Theory

Actuator disc theory states that, in a steady current, the mass flow rate must be conserved. Hence, the stream-wise velocity at the disc, u_t , can be determined through the relationship

$$u_t = (1-a)u_\infty,\tag{1}$$

where u_{∞} is the free stream velocity, and *a* is the axial induction factor [6]. Using momentum theory it is then possible to formulate expressions for the thrust, *T*, on and power, *P*, generated by the disc as functions of the free stream velocity

$$T = \frac{1}{2}\rho C_t A u_\infty^2,\tag{2}$$

201

$$P = \frac{1}{2}\rho C_p A u_\infty^3,\tag{3}$$

where A is the area of the disc and C_t and C_p are the thrust and power coefficients respectively, where

$$C_t = 4a(1-a),$$
 (4)

204

$$C_p = 4a(1-a)^2.$$
 (5)

Actuator disc methods are common in numerical models due to their sim-205 plicity, requiring only knowledge of the thrust coefficient and the free-stream 206 velocity. In this study, however, the turbine methodology is required to work 207 in transient flows such as those experienced in wave-driven environments, and 208 hence the free stream velocity is not known a priori. Therefore, the actuator 209 disc methodology is reverse engineered based on the known velocity at the tur-210 bine in order to estimate the instantaneous free stream velocity and the thrust 211 on the disc (explained further in Section 3.4). 212

²¹³ 3.2. Weighting Function

The first stage in the turbine model is the calculation of the weighting function (or field), W, which determines the contribution of each cell in the computational domain to the local flow velocity at the turbine (W also determines the distribution of the thrust force on the turbine and the corresponding distributed momentum sink (see Section 3.4)).

In this study, actuator disc theory is used to represent a HATT and so a cylindrical region is selected to represent the turbine. The cylinder has: a radius, R, equal to the radius of the swept area of the turbine blades, and; an axis coincident with that of the turbine. At each time step, all cells in the computational domain are evaluated to find the distance between their centre and the central line of this 'turbine region', i.e. the turbine axis. For a turbine axis parallel to the global x-axis,

$$\mathbf{dx} = \mathbf{x}_{cell} - \mathbf{x}_{hub} = (dx, dy, dz),\tag{6}$$

where \mathbf{x}_{cell} and \mathbf{x}_{hub} are the coordinates of the cell centre and the hub position of the turbine respectively. The *x*-component, dx, corresponds to the axial distance from the turbine plane and the axial width of the turbine is $2N\sigma$ (Figure 1). The radial components of the cylindrical region, dy and dz, are used to define another vector

$$\mathbf{r} = (0, dy, dz),\tag{7}$$

whose length, $|\mathbf{r}|$, determines the radial distance from the turbine axis. The edge of the turbine will then be located on the line $|\mathbf{r}| = R$. Consequently, the turbine region is made up of cells that have r values that fall within the range $R_{hub} \leq |r| \leq R$, where R_{hub} is the hub radius (Figure 1a).

In order to ensure mathematically smooth values for the calculated local 235 flow speed (and thrust forces) through time, for the general case in which the 236 motion of the turbine is not concurrent with the motion of the mesh cells, the 237 turbine region is given a finite width. The width is defined as $2N\sigma$ (N σ either 238 side of the centreline in the axial direction), where σ is the Gaussian root mean 239 square width as shown in Figure 1b and N is a user-defined coefficient to limit 240 the width of the turbine region (set by default to 2 according to the sensitivity 241 analysis in Section 4.2). A Gaussian weighting, is then determined for each cell 242 in the computational domain 243

$$W = \begin{cases} \frac{1}{\sigma\sqrt{2\pi}} \exp\left(-\frac{dx^2}{2\sigma^2}\right) & \text{if } |dx| \le N\sigma \text{ and } R_{hub} \le |\mathbf{r}| \le R, \\ 0 & \text{otherwise,} \end{cases}$$
(8)

with cells closer to the central plane of the turbine region having the largest 244 weights and, therefore, contributions in the proceeding calculations. Using this 245 method, any number of turbine regions can be represented simultaneously with-246 out a significant increase in computational effort. The sensitivity of the model 247 to various parameters is discussed in Section 4. Note that, for simplicity, the 248 presented model considers the weighting inside the turbine region to be uni-249 form in the radial direction and for all cells outside of the turbine region the 250 contribution to the local velocity is zero. 251

252 3.3. Orientation

For generality, the turbine model has been developed to allow the turbine to be placed in any orientation relative to the coordinate system of the computational domain. This is achieved via the orientation matrix, \mathbf{Q}_0 , defined as

$$\mathbf{Q}_{0}(\alpha,\beta,\gamma) = \mathbf{R}_{x}\left(\frac{\alpha}{2}\right)\mathbf{R}_{y}\left(\frac{\beta}{2}\right)\mathbf{R}_{z}\left(\gamma\right)\mathbf{R}_{y}\left(\frac{\beta}{2}\right)\mathbf{R}_{x}\left(\frac{\alpha}{2}\right),\tag{9}$$



Figure 1: Schematic representation of a) the 'turbine region' and b) the Gaussian weighting function used in the turbine model.

where \mathbf{R}_x , \mathbf{R}_y , \mathbf{R}_z are matrices defining a rotation about the global x, y and zaxes respectively. These are defined as

$$\mathbf{R}_{x} = \begin{pmatrix} 1 & 0 & 0\\ 0 & \cos\left(\alpha\right) & -\sin\left(\alpha\right)\\ 0 & \sin\left(\alpha\right) & \cos\left(\alpha\right) \end{pmatrix},\tag{10}$$

259

$$\mathbf{R}_{y} = \begin{pmatrix} \cos\left(\beta\right) & 0 & \sin\left(\beta\right) \\ 0 & 1 & 0 \\ -\sin\left(\beta\right) & 0 & \cos\left(\beta\right) \end{pmatrix},\tag{11}$$

260

$$\mathbf{R}_{z} = \begin{pmatrix} \cos\left(\gamma\right) & -\sin\left(\gamma\right) & 0\\ \sin\left(\gamma\right) & \cos\left(\gamma\right) & 0\\ 0 & 0 & 1 \end{pmatrix}, \tag{12}$$

where α , β and γ are the angles of roll, pitch and yaw respectively.

A new coordinate system, based on the orientation matrix, is then generated and each cell in the domain is assigned a new set of coordinates in the rotated system according to

$$\mathbf{dx}' = (dx', dy', dz') = \mathbf{Q}_0^T \cdot \mathbf{dx}$$
(13)

The turbine weighting is then calculated (as described in Section 3.2) but with $d\mathbf{x}$ replaced by $d\mathbf{x}'$.

267 3.4. Free Stream Velocity Calculation

As described in Section 3.1, the free stream velocity, u_{∞} , is required to calculate the thrust on, and power generated by, the turbine. However, in general and particularly in wave and current cases, u_{∞} is time-varying and is not known in advance. Therefore, a method to determine the instantaneous free stream velocity from the known velocity field, local to the turbine region, has been developed. Using the weighting function, W, as described in Section 3.2, the weighted average relative velocity, \mathbf{u}_{av} in the turbine region can be calculated using

$$\mathbf{u}_{av} = \left(\frac{1}{V}\sum_{i=1}^{N}\mathbf{u}_{i}V_{i}W_{i}\right) - \mathbf{v}_{hub},\tag{14}$$

where N is the total number of cells in the turbine region, \mathbf{v}_{hub} is the velocity of the turbine and V is the total weighted volume of the turbine region based on the volume of each cell, V_i ,

$$V = \sum_{i=1}^{N} V_i W_i. \tag{15}$$

The local speed in the axial direction, u_t , is then determined using

$$u_t = |\mathbf{u}_{av} \cdot \mathbf{x}_{axis}|,\tag{16}$$

280 where

$$\mathbf{x}_{axis} = \mathbf{Q}_0^T \cdot \hat{\mathbf{x}},\tag{17}$$

is a unit vector parallel to the axis of the turbine and $\hat{\mathbf{x}}$ is a unit vector in the global x-direction. For a turbine with known axial induction factor, a, or thrust coefficient, C_t , the instantaneous free stream velocity is then calculated using a rearrangement of equation (1).

The instantaneous thrust on the turbine, T, and the instantaneous power generated, P, can then be calculated using equations (2) and (3) respectively.

287 3.5. Update Momentum Equation

Assuming laminar flow and neglecting surface tension, the incompressible (unsteady) RANS equations take the following form

$$\frac{\partial(\mathbf{u})}{\partial t} + \nabla \cdot (\mathbf{u}\mathbf{u}) = -\nabla \frac{p}{\rho} + \nabla^2(\nu \mathbf{u}) + \mathbf{g} + \mathbf{T},$$
(18)

290

$$\nabla \cdot \mathbf{u} = 0, \tag{19}$$

where p is the pressure, ρ is the fluid density, $\mathbf{u} = (u, v, w)$ is the fluid velocity, ν is the kinematic fluid viscosity, \mathbf{g} is acceleration due to gravity and \mathbf{T} is the momentum sink due to the presence of the turbine.

To achieve coupling between the calculated thrust on the turbine, T, and the associated momentum sink in the fluid, \mathbf{T} , the thrust force per unit mass is distributed across the turbine region using the weighting function, W, (from Section 3.2) such that

$$\mathbf{T}_{i} = \pm \frac{TV_{i}W_{i}}{\rho V} \mathbf{x}_{axis},\tag{20}$$

where \pm is positive for flow in the direction of the turbine axis or negative for reversed flow.

The governing equations for the fluid (equation 18) can then be solved for the next time step (transient) or iteration (steady-state) and the processes described in Sections 3.2-3.5 repeated until the maximum time (transient) or convergence (steady-state) is reached.

4. Steady-State Analysis

In this section, the sensitivity of the turbine model, to the various implementation parameters discussed above, is demonstrated using the single phase, steady-state solver simpleFoam. For this analysis, a single static turbine (i.e. $|\mathbf{v}_{hub}| = 0$) is considered, in isolation. The objective here is to demonstrate the robustness of the model as well as verify that the model returns the expected results for idealised cases.

The modelling technique described in Section 3, relies fundamentally on predicting the free stream velocity, u_{∞} , from the local velocity in the turbine region, u_t . Hence, to quantify the accuracy of the model and assess the sensitivity of the approach to the key parameters, comparisons are made between the predicted value of u_{∞} and the user defined value at the inlet. The convergence criteria are kept constant throughout this analysis and are set to have maximum residual values of 10^{-4} and 10^{-5} for the pressure and velocity fields respectively.

318 4.1. Initial Setup

Unless stated otherwise, the sensitivity test cases use a turbine model with a radius of 2 m (the hub radius is set to zero), a $C_t = 0.9$, N = 2, and $\sigma = 0.15$. The prescribed free stream velocity is 1 m s^{-1} .

The required cell size around the turbine was evaluated using a mesh con-322 vergence study. The initial numerical domain was a $200 \times 20 \times 20$ m cuboid 323 consisting of cubic cells (side length = 0.5 m) and a 5 m cubic region around the 324 turbine, refined by one level using the octree refinement strategy [9]. The re-325 finement in the turbine region was then incrementally increased until the value 326 of the predicted free stream velocity changed by less than 0.1% between meshes. 327 This was found to occur for cells 3.125 cm in length (4 levels of refinement). All 328 remaining simulations, in the sensitivity analysis, use cells of this size in the 329 turbine region unless otherwise stated. 330

To optimise the dimensions of the computational domain, with respect to 331 a reduction in both the blockage effects arising from the boundaries and the 332 required computational effort, a series of tests focusing on the inlet, outlet and 333 side wall distances were performed. By incrementally increasing one of these 334 distances (whilst fixing the other two), the minimum distance from the turbine 335 was determined based on convergence of the predicted free stream velocity. By 336 applying this method to all boundaries, a $100 \times 100 \times 100$ m numerical domain 337 was selected for the sensitivity analysis, running from -50 m to 50 m in each 338 dimension, with the centre of the turbine located at the origin (Figure 2). 339



Figure 2: Sketch of the numerical domain used for the steady-state sensitivity analysis in a) the x - y plane at z = 0, and b) the y - z plane at x = 0, with the turbine region indicated in red.

The side walls of the domain are considered to be solid and have no-slip boundary conditions applied to them, the inlet boundary has the free stream velocity prescribed as a boundary condition and the outlet has zero gradient conditions to represent flow leaving the domain. The initial condition is the prescribed free stream velocity used at the inlet, and turbulence modelling has not been considered in this analysis.

346 4.2. Width Parameters

Actuator theory is based on an infinitesimally thin disc [11, 18] and therefore, 347 in the present model, a delta function to describe the turbine width would 348 likely give the most accurate solution. This would be possible in finite volume 349 350 methods, if the turbine was fixed and aligned perfectly with the cell centres (or faces). However, in the present study, the turbine model is coupled to a floating 351 structure which (as well as having arbitrary mesh motion and structure), in 352 general, has arbitrary alignment with the mesh. It is, therefore, necessary to 353 assign a finite region in which to ascertain the local flow velocity and apply 354 the corresponding momentum sink, as an infinitesimally thin region would not 355 perform well in cases (or time steps) in which there is a misalignment with the 356 computational mesh. In this section, the sensitivity of the model to the width 357 of this finite turbine region, $2N\sigma$, is considered. 358

A series of simulations were performed using different combinations of Nand σ and, using the converged solution for u_{∞} , the error [%] was calculated with respect to the prescribed inlet flow speed (1 m s^{-1}) .

Figure 3a shows the error, as a function of $\sigma\sqrt{N}$, for the case in which the central plane of the turbine is coincident with cell faces. For higher values



Figure 3: Error [%] in u_{∞} as a function of $\sigma\sqrt{N}$, when the turbine is aligned with a) cell faces, b) cell centres. Values which follow a log law (--) are indicated as circles (o) otherwise they are represented by crosses (\times). Also shown (c) is the difference [%] between the solutions obtained with the turbine aligned with cell faces and with cell centres.

of $\sigma\sqrt{N}$ (marked with o), as the width moves towards the ideal case of an 364 infinitesimally thin disc, the error decreases with a logarithmic trend (indicated 365 by the dashed line (--). However, below a certain 'cut-off' the logarithmic 366 trend breaks down (\mathbf{x}) ; the error initially decreases, before rapidly increasing. 367 The rapid increase in error at low $\sigma \sqrt{N}$ values is a consequence of the total 368 width of the turbine region approaching the width of a single cell. Based on 369 this, it seems that the optimal turbine width would be the value at which this 370 cut-off occurs, i.e. small enough to minimise the error, but large enough that 371 the results lie in the well-behaved, logarithmic region. For the combination 372 of mesh, turbine and flow speed used here, the cut-off value was found to be 373 $\sigma\sqrt{N} = 0.07$, with an error of 1% or less for $0.07 \le \sigma\sqrt{N} \le 0.11$. 374

In general, the turbine's central plane will be positioned arbitrarily relative

to each mesh cell and hence the solution must be independent of this parameter. 376 Figure 3b shows the error for the case in which the central plane of the turbine 371 is coincident with the mesh cell centres. With respect to the staggering of 378 the mesh cells relative to the turbine central plane, this represents the most 379 extreme alternative to the case used above which was coincident with the mesh 380 faces. From Figure 3b, it can be seen that the error for a turbine central plane 381 coincident with the cell centres has the same trend as that for that coincident 382 with the cell faces (Figure 3a). 383

The difference between the two solutions is presented in Figure 3c. For low 384 values of $\sigma\sqrt{N}$ there are unacceptable differences between the solutions; it is 385 anticipated that, in the general case of a turbine moving arbitrarily through 386 the mesh, these would generate unphysical fluctuations in critical values (e.g. 387 the thrust on the turbine). Although small differences can be observed around 388 $\sigma\sqrt{N} = 0.1$, the differences between the two solutions are much smaller in the 389 logarithmic region. This is due to the increased turbine width distributing the 390 weighting over more cells and reducing the sensitivity to single values (including 391 the difference between coincidence with a cell centre or a cell face). 392

The results presented in Figure 3 imply that the model is not overly sensitive 393 to the coincidence of the mesh cells and the turbine central plane, provided $\sigma \sqrt{N}$ 394 is reasonably large. This is essential for the model to be successful in a moving 395 mesh simulation. However, the results also demonstrate that for high levels 396 of accuracy the turbine region, i.e. $\sigma\sqrt{N}$, should be kept relatively small. As 397 a compromise, in this particular case, N and σ have been chosen to achieve 398 $\sigma\sqrt{N} = 0.11$ giving an error of around 1%, whilst maintaining a solution that 399 is suitably independent of the coincidence of the mesh cells and turbine central 400 plane. 401

402 4.3. Mesh Dependency

As mentioned above, the error in the predicted value of u_{∞} appears to have some mesh dependence at low $\sigma\sqrt{N}$ values. Consequently, further simulations were performed, with varying σ and N, with the mesh in the region of the turbine one octree level finer (1.5625 cm) or one level coarser (6.25 cm) compared to the mesh used in Section 4.2 (see Table 1 for details).

Figure 4 shows the error [%], as a function of $\sigma\sqrt{N}$, for each of the three mesh resolutions (original (\circ), finer (\triangle) and coarser (\diamond)). The dashed lines represent the logarithmic trends of the mesh in the corresponding colour. The

Table 1: Mesh resolution, aspect ratio, octree level and total size used in each of the steady-state, static cases.

Mesh	Background		Refined	Total Cells	
	$\Delta x \ [m]$	AR	Oct. lvl.	$\Delta x \ [m]$	
Coarse	0.5	1	3	0.0625	8.5M
Medium	0.5	1	4	0.0313	12.2M
Fine	0.5	1	5	0.0156	$40.7 \mathrm{M}$



Figure 4: Comparison of the error [%] as a function of $\sigma \sqrt{N}$ for three different mesh resolutions: $\Delta x = 6.25 \text{ cm} (\diamond)$, $3.125 \text{ cm} (\diamond)$ and $1.5625 \text{ cm} (\triangle)$. The lines represent the logarithmic error trend for each case dash-dotted, solid, and dashed, respectively.

gradient of the trend lines decreases with increasing resolution, as does the cut-411 off width, C, defining the end of the logarithmic region, i.e. for finer meshes, the 412 logarithmic region holds for much lower $\sigma \sqrt{N}$ and so higher accuracies can be 413 achieved before the model becomes too sensitive to motion through the mesh. 414 For higher $\sigma \sqrt{N}$ the solutions from the three meshes are very similar, indicating 415 mesh independence. It is unclear whether the logarithmic regions continue for 416 much higher values of $\sigma\sqrt{N}$, but it is unlikely that a width greater than those 417 considered here would be beneficial due to the increased error. 418

Based on the logarithmic trends observed in Figure 4, the error in the predicted value of u_{∞} takes the form

$$\mathcal{E}(\%) = \mathcal{A}\ln\left(\sigma\sqrt{N}\right) + \mathcal{B}, \quad \text{if } \sigma\sqrt{N} \ge \mathcal{C}$$
(21)

421 where \mathcal{A}, \mathcal{B} and \mathcal{C} are all functions of mesh resolution Δx .

Figure 5 shows that \mathcal{A} and \mathcal{C} (and to a reasonable degree \mathcal{B}) are linear functions of mesh resolution Δx , and for this case

 $\mathcal{A} = 5.0807\Delta x + 0.288, \quad \mathcal{B} = 4.0155\Delta x + 1.5418, \quad \mathcal{C} = 2.56\Delta x - 0.01.$ (22)

The coefficients in equation (22) are likely to be functions of the turbine diameter (4 m) and the incident flow speed (1 m s^{-1}) , however, by combining equations



Figure 5: Properties of the logarithmic error trend as a function of mesh resolution: a) gradient, b) intercept, and c) the minimum $\sigma\sqrt{N}$ cut-off value for which the trend holds.

(21) and (22) the error in this case could be estimated for any given mesh discretisation, allowing a suitable value of $\sigma\sqrt{N}$ to be chosen.

428 4.4. Flow Speed

⁴²⁹ Over a tidal cycle, a turbine will experience a wide range of flow speeds; ⁴³⁰ furthermore, for floating tidal energy applications the turbine will be subject ⁴³¹ to a combination of both tidal currents and waves, i.e. oscillatory flow. Con-⁴³² sequently, in order to model a full floating tidal energy concept, in realistic ⁴³³ conditions, it is vital that the performance of the turbine model is not overly ⁴³⁴ sensitive to the flow speed.

To assess the performance of the present model as a function of free stream 435 velocity, u_{∞} , a series of simulations were run with different prescribed flow 436 speeds (in the range $0.25 - 4 \,\mathrm{m \, s^{-1}}$). Figure 6a presents the predicted u_{∞} values 437 against the prescribed inlet velocities, with the red, dotted line representing 438 perfect prediction. The predicted and prescribed values generally agree very 439 well, although as the flow speed increases the deviation does appear to increase. 440 Considering Figure 6b (which shows the error as a function of prescribed in-441 let speed), it is clear, however, that the relative error remains very similar 442 throughout, i.e. $\approx 0.95\%$ for all of the 16 flow speeds tested. It can therefore 443 be concluded that, the model developed here performs equally well over the 444 required range of incident flow speeds. 445

446 4.5. Turbine Characteristics

So far in this section, the turbine characteristics have been fixed to represent a generic turbine with a radius of 2 m and a thrust coefficient of 0.9. In realistic applications, these parameters will be determined by the turbine manufacturer and, although it may be constant during operational conditions, the thrust coefficient could potentially change in order to produce favourable output characteristics or reduce the chance of damage to the generator at high flow



Figure 6: Sensitivity of the model to flow speed: a) predicted free stream value as a function of the prescribed value at the inlet, and b) the relative error as a function of prescribed flow speed.

speeds. Hence, for completeness, and generality, the performance of the model
is assessed for turbines of different thrust coefficients, utilising the same mesh
and simulation setup as presented earlier.

Figure 7 shows the error [%] as a function of thrust coefficient, C_t . In this case: For low thrust coefficients ($C_t < 0.65$) the free stream velocity is slightly under-predicted; for high thrust coefficients $C_t \ge 0.65$ the free stream velocity is over-predicted. Further work is required to understand this behaviour for different turbine characteristics, flow speeds and domain sizes but it appears that, for all except the very highest C_t values, the predicted free-stream velocity is well within 1% of the true value.

⁴⁶³ 5. Prescribed Motion Cases

The aim of this work is to develop a turbine methodology that can be used 464 in the simulation of entire floating tidal stream systems. In Section 4, the 465 methodology is shown to predict with good accuracy the free stream velocity, 466 and hence the thrust, in the case of a static turbine. However, when simulat-467 ing the complete coupled system, the movement of the device (in any of six 468 degrees of freedom) leads to a time-varying turbine position with arbitrary lo-469 cation and alignment with the numerical grid. This prevents the use of highly 470 contrived meshes designed solely to capture the turbine well [7, 12, 24] and 471 requires a methodology capable of seamlessly transitioning through the mesh 472 without causing numerical instabilities. In this section the ability of the present 473 method, to meet this requirement, is demonstrated via a series of test cases in 474 which the turbine is given prescribed motion through the computational domain. 475



Figure 7: Sensitivity of the model accuracy to turbine thrust coefficient, C_t .

In each case, a turbine with R = 2 m, $C_t = 0.9$, $N = 2 \text{ and } \sigma = 0.15$ is moved 476 with prescribed velocity, \mathbf{v} , through initially still water and the predicted free 477 stream velocity is again compared to the true u_{∞} . The tests were run for 478 60 s using the modified transient solver pisoFoam (see Section 3) and the same 479 $100 \times 100 \times 100$ m domain as that described in Section 4 (see Figure 2). Only 480 the refined region in the path of the turbine, its initial location/orientation and 481 prescribed velocity vary in each case. The boundary conditions are the same as 482 described in Section 4 (with inflow speed of $0 \,\mathrm{ms}^{-1}$), the initial conditions are 483 zero flow conditions, and turbulence modelling has not been considered in this 484 section. 485

In all cases, the speed of the turbine is ramped up to avoid effects arising from instantaneous movement of the turbine. The ramp up is described by the sinusoidal function

$$\mathbf{v}_{hub} = \begin{cases} \frac{1}{2} \mathbf{v} \left[1 - \cos\left(\frac{\pi}{t_{ramp}} t\right) \right] & \text{if } t < t_{ramp} \\ \mathbf{v} & \text{if } t \ge t_{ramp}, \end{cases}$$
(23)

where t_{ramp} is the ramp up time (set to 20 s in this work) and the position of the turbine is updated based on the integral of this function.

⁴⁹¹ 5.1. Constant Linear Velocity

In the first two test cases, the turbine is given a constant velocity through the mesh, $|\mathbf{v}_{hub}| = 1 \,\mathrm{ms}^{-1}$. These cases are considered to be equivalent to the idea of a physical towing tank and it is anticipated that the turbine behaves the same as if it were fixed in uniform flow with velocity equal to the prescribed motion, i.e. the relative flow over the turbine is the same.



Figure 8: Time series of the prediction of u_{∞} in the aligned $(\cdot \cdot \cdot)$, misaligned $(- \cdot -)$ and angular (—) prescribed velocity cases, along with the prescribed velocity (--). Both the full time series (a) and a magnified view (b) are presented.

Table 2: Initial turbine position, orientation and refined region for each of the prescribed motion cases. The mesh resolution is given by the 'coarse' mesh in Table 1, and the refined region is [-2.5, 2.5] in the z direction in all cases.

Case	Init. Hub Pos.			Refi	ned Region	# Cells	Timestep	
	x	y	γ	x	y	γ		$\Delta t \ [s]$
Aligned	30	0	0°	[-31, 31]	[-2.5, 2.5]	0°	14.6M	0.01
Misaligned	18	-18	45°	[-22, 40]	[-20, 20]	45°	14.6M	0.01
Angular	0	-8	0°	[-13, 13]	[-13, 13]	0°	$19.1 \mathrm{M}$	0.01

The first test demonstrates the case of the turbine moving parallel to the x-axis, i.e. aligned with the mesh. At time t = 0 s the centre of the turbine is located at $\mathbf{x}_{hub} = (30, 0, 0)$ m and, after the period of ramp up, the turbine moves with the constant prescribed velocity, $\mathbf{v}_{hub} = (-1, 0, 0)$ ms⁻¹. The mesh in the region along the path of the turbine (Table 2) is refined by three octree levels ($\Delta x = 0.0625$ m), which (based on the information in Section 4) gives an anticipated error in the predicted free stream velocity of approximately 1%.

Figure 8a presents a time series of the predicted free stream velocity, u_{∞} , 504 for the aligned case $(\cdot \cdot \cdot)$ and the final, prescribed speed, i.e. the true solution 505 (--). The initial ramp up of the turbine velocity can be observed, along with 506 an over-shoot (~ 6%) as the ramp up period ends. After this, the prediction 507 converges towards the anticipated solution with an error of approximately 1%508 (observed at time t = 60 s). Crucially, the prediction of u_{∞} is relatively smooth, 509 indicating that the present methodology works well for turbines moving parallel 510 to the axes of the mesh. There are some very small fluctuations ($\sim 0.001 \,\mathrm{ms}^{-1}$) 511 in the prediction that can be observed when considered more closely (Figure 8b), 512 which are likely due to the instantaneous position of the turbine (see Section 4.2) 513

⁵¹⁴ but these are considered to be negligible and are not expected adversely to affect
 ⁵¹⁵ the stability of the simulation.

The second case considers turbine motion that is not aligned with the axes of 516 the mesh. This is achieved by rotating the turbine axis by 45° and prescribing a 517 constant velocity $\mathbf{v}_{hub} = (-0.707, 0.707, 0) \,\mathrm{ms}^{-1}$. The turbine is initially located 518 at $\mathbf{x} = (18, -18, 0)$ and the mesh along the path of the turbine (Table 2) is again 519 refined by three octree levels (again with an anticipated error of 1%). The 520 predicted value of u_{∞} , in this case, is presented in Figure 8a $(-\cdot -)$. The time 521 series is very similar to that observed for the aligned case; after the initial ramp 522 up period, the prediction overshoots before converging to 1% of the prescribed 523 speed by t = 60 s. The error is marginally larger than that observed in the 524 aligned case but, interestingly, the fluctuations in the prediction are smaller 525 (Figure 8b). This is thought to be caused by the 'mis-alignment' of the turbine: 526 In the aligned case, the edge of the turbine region crosses cell faces at all points 527 simultaneously; hence, any slight differences between the cells also contribute to 528 the solution simultaneously resulting in a more noticeable change. In the non-529 aligned case, the edge of the turbine region crosses the cell faces arbitrarily and 530 hence the differences contribute asynchronously resulting in lower fluctuations. 531

532 5.2. Constant Angular Velocity

In Section 5.1 the present turbine methodology is shown to be robust and accurate, when moving at a constant linear velocity (either aligned and misaligned with the computational mesh). Floating tidal stream devices, however, are capable of moving in all six degrees of freedom and so, it is crucial that the methodology can also accommodate rotational motion through the mesh.

The third prescribed motion test case considers the turbine rotating about 538 the z-axis with a constant angular velocity, $\omega = 0.125 \,\mathrm{rad}\,\mathrm{s}^{-1}$. The turbine's 539 velocity is given by the instantaneous tangential velocity (at the turbine hub), 540 $\mathbf{v}_{hub} = \omega \mathcal{R}$, where \mathcal{R} is the orbital radius. At each time step the value of γ 541 has been updated (relative to the centre of the orbit), and \mathbf{R}_z (equation 12) is 542 applied an additional time in equations (13) and (17) to capture the rotation of 543 the turbine. In this case $\mathcal{R} = 8 \,\mathrm{m}$ and so $|\mathbf{v}_{hub}| = 1 \,\mathrm{ms}^{-1}$. One complete orbit 544 takes approximately 60 s, ensuring the turbine region does not interact with the 545 wake from the previous orbit. The mesh along the path of the turbine (Table 2) 546 is refined to the same discretisation as in the linear velocity cases. 547

The predicted free stream velocity in the rotating case is presented in Fig-548 ure 8a (—). The results show the same trend as in the linear velocity cases: An 549 initial over-shoot in the prediction after the ramp up period, before converging 550 to within 2% of the expected solution. This error is slightly larger than in the 551 linear velocity cases, however, this might be anticipated as the underlying the-552 ory behind the expected solution is based on uniform flow across the turbine 553 554 (and this is not true in this case). Crucially, again, there are only negligible fluctuations in the predicted solution (Figure 8b) indicating that the present 555 methodology performs well even with arbitrary mesh alignment and rotational 556 motion through the computational mesh. 557



Figure 9: Numerical domain used for the two-phase simulations, in the x - z (a), and x - y (b) planes. Information regarding the mesh resolution is indicated in red, with double headed arrows representing mesh grading. The green shading indicates the refined region.

558 6. Velocity Deficit Validation

In Sections 4 and 5 the numerical model is shown to be robust and capable of 559 capturing turbine loads when moving through a mesh, which is the primary mo-560 tivation for the model. However, a secondary objective is to determine whether 561 the turbine models influence on the fluid is captured accurately. Therefore, in 562 this section the model is validated against existing experimental data for the 563 velocity deficit behind a porous disc [31, 14]. These experiments were conducted 564 in the Chilworth research laboratory flume at the University of Southampton. 565 which is 21 m in length, 1.35 m wide and used a nominal water depth of 0.3 m566 [31]. Small scale discs (\emptyset 0.1 m, 0.001 m width) of varying porosity (Ct = 0.61, 567 0.86 and 0.94) were evaluated, with wake profile measurements taken at a point 568 location (varied between runs) using an Acoustic Doppler Velocimeter (ADV). 569

The interFoam solver (see Section 3) coupled with the developed turbine 570 model is used to simulate the problem. Current speeds of $0.2487 \,\mathrm{ms}^{-1}$ are 571 generated using the expression based boundary condition and relaxation zone 572 technique provided as part of the waves2Foam toolbox [17]. The $k - \omega$ SST 573 turbulence closure scheme [28] is used to model the turbulent effects. For com-574 putational efficiency, the numerical model simulates one half of the flume (and 575 disc), assuming that the flow is symmetric at the y = 0 plane. The water depth 576 is set to 0.3 m ($-0.3 \le z \le 0.3$) and the tank width is 0.675 m ($-0.675 \le y \le 0$), 577 consistent with the experiments [31]. The simulated length of the tank is set to 578 $8 \text{ m} (-2 \le x \le 6)$, to accommodate an inlet region $(-2 \le x \le 0)$, a working 579 region 30D in length $(0 \le x \le 3)$ and a relaxation zone $(3 \le x \le 6)$. The initial 580 mesh is designed such that the Aspect Ratio (AR) is set to 1 in the working 581 region, with a mesh resolution of $\Delta x = 0.01$ m (Figure 9). Mesh grading is used 582 to reduce computational cost in all directions: $x \ge 3$, $y \le -0.1$ and $z \ge 0.1$. 583



Figure 10: Comparison of experimental [31] (o) and numerical predictions (——) of centreline, horizontal velocity deficit profiles for $C_t = 0.61$ (a), 0.86 (b) and 0.94 (c).

Two levels of additional octree refinement [10] are used in the region of the wake of the turbine $(-2 \le x \le 3, \text{ radius } 0.1 \text{ m}, \Delta x = 0.0025 \text{ m}).$

The disc is centred at x = 0, z = -0.15, with R = 0.05 m, $\sigma = 0.005$ 586 and N = 2, which gives an expected error of approximately 1% based on the 587 results in Section 4, and each case is run for 120s of simulation time. The inlet 588 and outlet boundaries for velocity are both set to the prescribed free stream 589 velocity $(0.2487 \,\mathrm{ms}^{-1})$ in the water phase $(0 \,\mathrm{ms}^{-1})$ in the air phase). The top 590 boundary is modelled as an atmosphere condition with a total pressure condition 591 applied. The bottom and side boundaries are considered to be walls and hence 592 are modelled with no-slip conditions. Wall functions are used for the turbulent 593 parameters at these boundaries and hence mesh refinement is applied adjacent to 594 these boundaries to achieve a suitable y^+ value ($y^+ \approx 40$). The inlet turbulent 595 conditions are determined based on an inlet turbulent intensity of 5%, with 596 zero gradient conditions applied at the outlet and atmosphere boundaries. The 597 initial conditions for velocity and turbulence parameters is set to the values 598 specified at the inlet. 599

Figure 10 presents a comparison of experimental (•) and numerical prediction (—) for the disc's centreline (y = 0 m, z = -0.15 m) velocity deficit profile as a function of diameters downstream, for $C_t = 0.61$ (a), 0.86 (b) and 0.94 (c). In all cases, the numerical predictions agree well with the experimental data. The near wake region ($x \leq 5D$) was observed to increase with thrust coefficient in the experimental data. The numerical model captures this effect due to thrust coefficient well, with progressively increasing velocity deficit: the



Figure 11: Comparison of experimental [12] (\circ) and numerical predictions (——) of vertical velocity deficit profiles for a $C_t = 0.86$ disc, at x = 4D (a), 7D (b), 11D (c), 15D (d) and 20D (e).

⁶⁰⁷ predictions of maximum velocity deficits are 0.58, 0.87 and 0.98 for $C_t = 0.6$, ⁶⁰⁸ 0.86 and 0.94, respectively, although it should be noted that these can not be ⁶⁰⁹ validated since the experimental campaign only considered positions for $x \ge 3D$.

A comparison of the experimental (•) and numerical predictions (-----) of 611 vertical profiles is presented in Figure 11 for the $C_t = 0.86$ at a number of 612 horizontal locations: x = 4D (a), 7D (b), 11D (c), 15D (d) and 20D (e). At 613 4D in the experimental data, there is a region of high velocity deficit, which 614 extends from $z/D \approx -2$ to $z/D \approx -1$, i.e. the position of the disc. This is 615 also observed in the numerical predictions, and the maximum occurs slightly 616 below the centreline of the disc at this location, which has also been observed in 617 previous CFD studies of the wake structure behind an actuator disc in a marine 618 environment [4, 34]. Moving further away from the disc, the experimental data 619 shows that this region reduces in magnitude and increases in height, which is 620 also captured by the numerical model. However, the maximum value gets lower 621 with increasing x in the numerical predictions, which although not obvious in the 622 point measurements presented in Figure 11, could be observed in spatial plots 623 presented by Myers and Bahaj [31]. In this work, the behaviour is more clearly 624 observed in spatial plots of the numerical data (Figure 12), and is consistent 625 for each of the discs considered. The spatial plots also show that the wake 626 distribution for $x \ge 8D = 0.8$ m is very similar for the three discs. This indicates 627 that the far wake structure is independent of the properties of the disc, and is 628 in-line with the observations of Myers and Bahaj [31]. 629

Overall, the numerical model captures velocity deficit to a similar standard as other numerical models [12, 4, 34], and distributions are comparable with experimental data [31]. Therefore, it is concluded that the model would be suitable for investigating both the effect of the turbine on a structures motion, and the implications for the fluid flow, in future work.



Figure 12: Velocity deficit contours for different turbine thrust coefficients: $C_t = 0.61$ (a), $C_t = 0.86$ (b) and $C_t = 0.94$ (c).

635 7. Conclusions

A new turbine model which can be used as a component of a framework 636 for simulating entire floating tidal systems has been presented. Analysis of 637 the new model in steady-state conditions showed that the prediction of the 638 free stream velocity could be replicated to within a 2% accuracy relative to 639 theoretical solutions, and this could be further reduced by tuning the width 640 parameter and mesh resolution. However, the key aspects of the model were 641 defined by the requirement to use the model for simulation of entire floating tidal 642 systems: the model has been shown to be insensitive to flow velocity, performs 643 well in any alignment with the mesh, and is capable of predicting the free stream 644 velocity while moving through the mesh under both linear and angular velocity. 645 These properties are crucial when simulating floating tidal systems, since such 646 systems will be required to survive in complex, non-linear environments driven 647 by strong wave-current interactions, requiring the turbine methodology to be 648 robust during changing flow velocity. Furthermore, the system will be capable 649 of moving in 6 degrees of freedom, and hence, the turbine will generally be 650 arbitrarily aligned with a mesh, and must be able to accurately predict the free 651 stream velocity under both linear and angular movement. 652

Following the success of the turbine model presented in this work, future research will focus on the development of a new coupled framework for simulating entire floating tidal systems, including integration of the present model.

656 Acknowledgements

The authors acknowledge that, this work was funded as part of Innovate UK 657 Project 102217 through the Energy Catalyst Early-stage Round 3. The code 658 presented here is available through the Collaborative Computational Project in 659 Wave Structure Interaction (CCP-WSI) [EP/M022382/1], which aims to bring 660 together the community of researchers, data, code and expertise within the area 661 of wave structure interaction (WSI) and has the objective of building an open-662 source, community-driven numerical wave tank (NWT) facility and capacity for 663 high quality research in an area of high importance (https://www.ccp-wsi. 664 ac.uk). 665

666 References

- [1] Abolghasemi, M., Piggott, M., Spinneken, J., Viré, A., C.J., C., Cram mond, S., 2016. Simulating tidal turbines with multi-scale mesh optimisa tion techniques. Journal of Fluids and Structures 66, 69–90.
- [2] Ammara, I., Leclerc, C., Masson, C., 2002. A viscous three-dimensional
 method for aerodynamic analysis of wind farms. J. Sol. Energ. Eng. 124,
 345–356.
- [3] Aubrun, S., Loyer, S., Hancock, P., Hayden, P., 2013. Wind turbine wake
 properties: Comparison between a non-rotating simplified wind turbine
 model and a rotating model. J. Wind Eng. Ind. Aerodyn. 120, 1–8.
- [4] Batten, W., Harrison, M., Bahaj, A., 2013. Accuracy of the actuator discRANS approach for predicting the performance and wake of tidal turbines.
 Philosophical Transactions of the Royal Society A 371.
- ⁶⁷⁹ [5] Blackmore, T., Batten, W., Bahaj, A., 2014. Influence of turbulence on
 ⁶⁸⁰ the wake of a marine current turbine simulator. Proc. R. Soc. A 470, 17.
- [6] Burton, T., Jenkins, N., Sharpe, D., Bossanyi, E., 2001. Wind Energy
 Handbook. 2 ed., John Wiley & Sons, Ltd.
- [7] Castellani, F., Vignaroli, A., 2013. An application of the actuator disc
 model for wind turbine wakes calculations. Appl. Energ. 101, 432–440.
- [8] Churchfield, M., Li, Y., Moriarty, P., 2013. A large-eddy simulation study
 of wake propagation and power production in an array of tidal-current
 turbines. Phil. Trans. R. Soc A 371.
- [9] Greaves, D., Borthwick, A., 1999. Hierarchical tree-based finite element
 mesh generation. Int. J. Numer. Meth. Eng. 45, 447–471.
- [10] Greaves, D.M., 2004. A quadtree adaptive method for simulating fluid flows
 with moving interfaces. Journal of Computational Physics 194, 35–56.

- [11] Hansen, M., 2000. Aerodynamics of wind turbines. 1 ed., James & James
 (Science Publishers) Ltd.
- [12] Harrison, M., Batten, W., Bahaj, A., 2010a. A blade element actuator disc
 approach applied to tidal stream turbines, in: Proceedings of OCEANS,
 IEEE, pp. 1–8.
- [13] Harrison, M., Batten, W., Bahaj, A., Myers, L., 2009. A comparison be tween CFD simulations and experiments for predicting the far wake of
 horizontal axis tidal turbines, in: Proceedings of the 8th European Wave
 and Tidal Energy Conference, Uppsala, Sweden. pp. 566–575.
- [14] Harrison, M., Batten, W., Myers, L., Bahaj, A., 2010b. Comparison be tween CFD simulations and experiments for predicting the far wake of
 horizontal axis tidal turbines. IET Renewable Power Generation 4, 613–
 627.
- [15] Holst, M., Dahlhaug, O., Faudot, C., 2014. CFD analysis of wave-induced
 loads on tidal turbine blades. IEEE J. Ocean. Eng. 40, 506–521.
- [16] Howard, R., Pereira, J., 2006. A study of wind turbine power generation
 and turbine/tower interaction using large eddy simulation. Wind Struct.
 9, 95–108.
- [17] Jacobsen, N.G., Fuhrman, D.R., Fredsøe, J., 2012. A wave generation
 toolbox for the open-source CFD library: OpenFOAM[®]. International
 Journal for Numerical Methods in Fluids 70, 1073–1088.
- [18] Johnson, B., Francis, J., Howe, J., Whitty, J., 2014. Computational actuator disc models for wind and tidal applications. J. Renew. Energ. 2014, 10.
- [19] Lawson, M., Li, Y., Sale, D., 2011. Development and verification of a computational fluid dynamics model of a horizontal-axis tidal current turbine,
 in: Proceedings of the 30th International Conference on Ocean, Offshore,
 and Arctic Engineering, Rotterdam, The Netherlands. pp. 1–10.
- [20] Leclerc, C., Masson, C., 2004. Toward blade-tip vortex simulation with an
 actuator-lifting surface model. AIAA Paper 2004-0667, 9.
- [21] Li, Y., Paik, K.J., Xing, T., Carrica, P., 2012. Dynamic overset CFD
 simulations of wind turbine aerodynamics. Renew. Energ. 37, 285–298.
- [22] Lignarolo, L., Ragni, D., Ferreira, C., van Bassel, G., 2016. Experimental
 comparison of a wind-turbine and of an actuator-disc near wake. J. Renew.
 Sustain. Energ. 8, 26.
- [23] Madsen, H., Bak, C., Døssing, M., Mikkelsen, R., Øye, S., 2010. Valida tion and modification of the Blade Element Momentum theory based on
 comparisons with actuator disc simulations. Wind Energ. 13, 373–389.

- [24] Malki, R., Williams, A., Croft, T., Togneri, M., Masters, I., 2013. A
 coupled blade element momentum computational fluid dynamics model
 for evaluating tidal stream turbine performance. Appl. Math. Model. 37,
 3006–3020.
- [25] Martínez, L., Leonardi, S., Churchfield, M., Moriarty, P., 2012. A com parison of actuator disk and actuator line wind turbine models and best
 practices for their use, in: 50th AIAA Aerospace Sciences Meeting, p. 13.
- ⁷³⁷ [26] Martínez-Tossas, L., Churchfield, M., Leonardi, S., 2015. Large eddy simulations of the flow past wind turbines: actuator line and disk modeling.
 ⁷³⁹ Wind Energy 18, 1047–1060.
- [27] Masters, I., Chapman, J., Willis, M., Orme, J., 2011. A robust blade
 element momentum theory model for tidal stream turbines including tip
 and hub loss corrections. J. Mar. Eng. Techno. 10, 25–35.
- [28] Menter, F.R., 1994. Two-equation eddy-viscosity turbulence models for
 engineering applications. AIAA Journal 32, 1598–1605.
- [29] Mikkelsen, R., 2003. Actuator disc methods applied to wind turbines. Ph.D.
 thesis. Technical University of Denmark.
- [30] Moriarty, P., Hansen, A., 2005. AeroDyn Theory Manual. Technical Report
 NREL/TP-500-36881. NREL.
- [31] Myers, L., Bahaj, A., 2010. Experimental analysis of the flow field around horizontal axis tidal turbines by use of scale mesh disk rotor simulators.
 Ocean Engineering 37, 218–227.
- [32] Naderi, S., Parvanehmasiha, S., Torabi, F., 2018. Modeling of horizontal axis wind turbine wakes in Horns Rev offshore wind farm using an improved actuator disc model coupled with computational fluid dynamic. Energy Conversion and Management 171, 953–968.
- [33] Naderi, S., Torabi, F., 2017. Numerical investigation of wake behind a
 HAWT using modified actuator disc method. Energy Conversion and Man agement 148, 1346–1357.
- ⁷⁵⁹ [34] Nguyen, V., Guillou, S., Thiébot, J., Cruz, A., 2016. Modelling turbulence
 ⁷⁶⁰ with an Actuator Disk representing a tidal turbine. Renewable Energy 97,
 ⁷⁶¹ 625–635.
- [35] Pintar, M., Kolios, A., 2013. Design of a novel experimental facility for
 testing of tidal arrays. Energies 6, 4117–4133.
- [36] Rajagopalan, R., Rickerl, T., 1990. Aerodynamic interference of vertical
 axis wind turbines. J. Propul. Power 6, 645–653.

- ⁷⁶⁶ [37] Rusche, H., 2002. Computational fluid dynamics of dispersed two-phase
 ⁷⁶⁷ flows at high phase fractions. Ph.D. thesis. Imperial College of Science,
 ⁷⁶⁸ Technology & Medicine.
- [38] Sanderse, B., van der Pijl, S., Korne, B., 2011. Review of computational
 fluid dynamics for wind turbine wake aerodynamics. Wind Energy 14,
 771 799–819.
- [39] Shen, W., Zhang, J., Søresen, J., 2005. Tip loss correction for
 actuator/Navier-Stokes computations. J. Sol. Energ. 127, 5.
- [40] Shen, W., Zhang, J., Søresen, J., 2009. The actuator surface model: A new
 Navier-Stokes based model for rotor computations. J. Sol. Energ. 131, 9.
- [41] Sørensen, J., Mikkelsen, R., Henningson, D., Ivanell, S., Sarmast, S. amd Andersen, S., 2015. Simulation of wind turbine wakes using the actuator line technique. Phil. Trans. R. Soc. A 373, 16.
- [42] Sørensen, J., Shen, W., 2002. Numerical modeling of wind turbine wakes.
 J. Fluid. Eng. 124, 393–399.
- [43] Sørensen, N., Michelsen, J., Schreck, S., 2002. Navier-Stokes predictions of
 the NREL phase VI rotor in the NASA Ames 80ft × 120ft wind tunnel.
 Wind Energ. 5, 151–169.
- [44] Svenning, E., 2010. Implementation of an actuator disk in OpenFOAM.
 Technical Report. Chalmers University of Technology.
- [45] Viré, A., Xiang, J., Piggott, M., Cotter, C., Pain, C., 2013. Towards
 the fully-coupled numerical modelling of floating wind turbines. Energy
 Procedia 35, 43–51.
- [46] Wang, Q., Zhou, H., Wan, D., 2012. Numerical simulation of wind turbine
 blade-tower interaction. J. Mar. Sci. Appli. 11, 321–327.
- [47] Watters, C., Masson, C., 2010. Modeling of lifting-device aerodynamics
 using the actuator surface concept. Int. J. Numer. Meth. Fluids 62, 1264–
 1298.
- [48] Weller, H., Greenshields, C., Bainbridge, W., Janssens, M., Santos, B.,
 2017. OpenFOAM 4.1. URL: https://openfoam.org/version/4-1/.
- [49] Wilson, J., Davis, C., Venayagamoorthy, S., Heyliger, P., 2015. Comparisons of horizontal-axis wind turbine wake interaction models. Journal of Solar Energy Engineering 137, 8.
- [50] Zahle, F., Søresen, N., Johansen, J., 2009. Wind turbine rotor-tower in teraction using an incompressible overset grid method. Wind Energ. 12,
 594-619.