

This is an Open Access document downloaded from ORCA, Cardiff University's institutional repository: <https://orca.cardiff.ac.uk/id/eprint/171779/>

This is the author's version of a work that was submitted to / accepted for publication.

Citation for final published version:

Hanousek, Nicolas , Ranabhat, Bikash, English, Aaron and Ahmadian, Reza 2024. A smoothed particle hydrodynamics method for vertical axis turbine design and assessment. *Journal of Hydrodynamics* 36 , pp. 991-1007. 10.1007/s42241-024-0074-y

Publishers page: <https://doi.org/10.1007/s42241-024-0074-y>

Please note:

Changes made as a result of publishing processes such as copy-editing, formatting and page numbers may not be reflected in this version. For the definitive version of this publication, please refer to the published source. You are advised to consult the publisher's version if you wish to cite this paper.

This version is being made available in accordance with publisher policies. See <http://orca.cf.ac.uk/policies.html> for usage policies. Copyright and moral rights for publications made available in ORCA are retained by the copyright holders.



A SMOOTHED PARTICLE HYDRODYNAMICS METHOD FOR VERTICAL AXIS TURBINE DESIGN AND ASSESSMENT

Nicolas Hanousek, HanousekN@Cardiff.ac.uk¹ **ORCID: 0000-0003-4458-2404**

Bikash Ranabhat, RanabhatB@Cardiff.ac.uk¹

Aaron English, aaronrobert.english@unipr.it² **ORCID: 0000-0001-9250-4182**

***Reza Ahmadian**, AhmadianR@Cardiff.ac.uk¹ **ORCID: 0000-0003-2665-4734**

*corresponding author

¹Hydro-environmental Research Centre, Cardiff University School of Engineering, Cardiff
CF24 3AA, United Kingdom

²Department of Engineering and Architecture, University of Parma, v.le Parco Area delle Scienze
181/A Parma, Italy, 43124

Keywords: Smoothed Particle Hydrodynamics, DualSPHysics, Vertical-axis Turbine, Hydrokinetic
Energy, Renewable Energy

ABSTRACT

With the global demand for more electricity, and for that electricity to be produced using low-carbon generation, a turbine was designed to extract energy from underutilised flows. The mechanism by which the turbine operates makes it highly demanding to represent using mesh-based numerical schemes, resulting in a need to investigate alternative methods. The smoothed particle hydrodynamics (SPH) software, DualSPHysics, utilising the Chrono solid body solver, was used to represent the turbine as a free body in a 2D environment allowing for evaluation of the free-spin velocity to be assessed. The aim of this being to ascertain the applicability of SPH to the modelling of vertical axis turbines with multiple moving parts, and also develop an understanding of the design itself. The model was found to compare favourably to lab results, showing that a vertical axis turbine may be represented in this fashion. The resilience of the device, a design driver – and previously untested mode, was assessed by considering post-damage scenarios. From this, future flume study and parallel numerical modelling can guide this or other vertical axis turbines towards improved performance.

HIGHLIGHTS

- Vertical axis turbine modelled in 2D SPH, without prescribed motion.
- Large shifting values (-50) and artificial viscosity required to prevent voids.
- Model convergence to particle spacing and kernel support $> O(2)$.
- Model was not sensitive to downstream boundary position.
- Hydrodynamically driven self start shown including damaged device cases.

ACKNOWLEDGEMENTS

This work was funded as part of the Water Informatics Science and Engineering Centre for Doctoral Training (WISE CDT) under a grant from the Engineering and Physical Sciences Research Council (EPSRC), grant number EP/L016214/1. We acknowledge the support of the Supercomputing Wales project, which is part-funded by the European Regional Development Fund (ERDF) via the Welsh Government.

1. INTRODUCTION

With 192 countries setting targets to reduce emissions and adapt to climate change impacts as part of the Paris Agreement, and a majority of those adopting more stringent measures since then, there is an increased need for low-carbon energy [1]. In the United Kingdom (UK) tidal stream has been estimated as having 95 terawatt-hours of energy potential per year, or 32 gigawatts of potential power [2], of which 15 gigawatts has been identified as practically available for deployment [3]. The majority of tidal stream turbine designs are horizontal axis with the same fundamental principles as wind turbines; these typically require large flow velocities (> 2.5 m/s) and deep water (> 2 m) which can be challenging to locate and effectively utilise [4]. The coastline of Great Britain for example rarely shows peak spring velocities in excess of 2 m/s, and the locations where velocities in this region can be seen are often not adjacent to population centres – as is shown in Fig. 1. The tidal predictability and energy profile make tidal stream a valuable component of future energy scenarios, further improving diversification of the energy mix [5]. Another potentially underutilised source of energy is the hydrokinetic potential of small-scale riverine flows. These exist all around the world, both in developed energy markets and in remote locations where grid connectivity has not yet been established or is not currently feasible. The global riverine energy resource from hydrokinetic devices has been estimated at approximately 58,000 terawatt-hours per year, with much of this potential being accessible to remote or isolated communities [6]. For this, design robustness, unit price, and the ability of the turbine to self-start are key, as the availability of complex maintenance methods may be reduced, to mitigate some of the key problems associated with small hydrokinetic turbines [7].

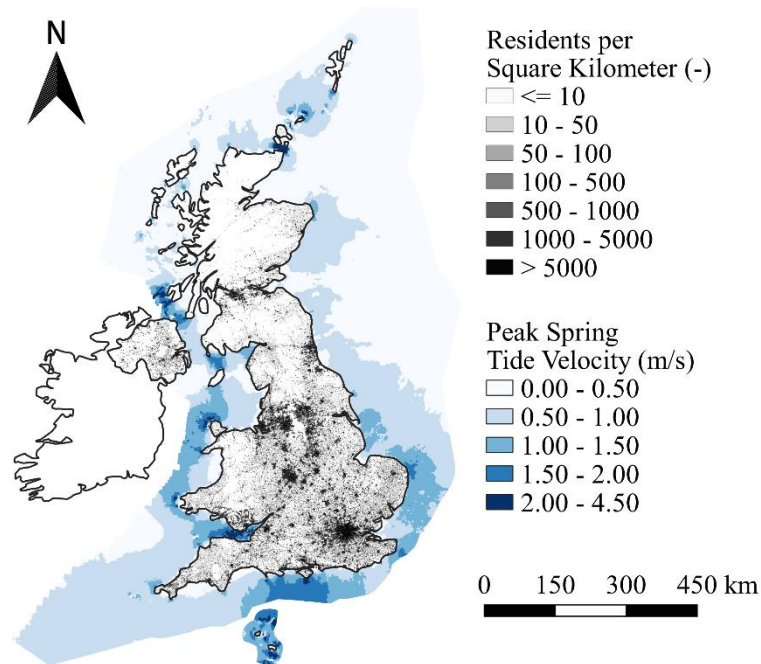


Fig. 1 Comparison of mean peak tidal velocity and population density around the UK, tidal stream data from the Association of British Ports [8], and population from UK census [9].

Vertical axis turbines offer several fundamental technical differences which allow users to tap into different resources beyond the scope of horizontal axis designs. In comparison with horizontal axis designs: vertical axis turbines can be operated in a wider range of depths (being manufactured at different heights or to operate both fully and partially submerged) and velocities; do not require yawing systems to best match the flow direction; are more resilient to skewed flows; often do not require complex components, and offer convenient options for generator connection [10]. The downsides of vertical axis designs are the lower performance and reduced control ability, two factors that in wind generation have made horizontal axis designs the status quo for large generators [11]. However, Kumar

et al. [12] note that the lower cut-in speed, low noise production and response to varying wind conditions make them suited for use in urban areas as small-scale units.

For deployment in tidal flows and riverine locations, a vertical axis turbine was developed at Cardiff University, namely ‘CarBine’ [13]. This scalable drag turbine takes the principles of a Savonius turbine and applies self-actuating hinges to maximise drag on the generating side, while minimising resistance on the returning side. The fundamental design comprises a central axle with hinged flaps mounted around it on notional arms. The motion of the flaps is restricted by the central axle and the axle of the other flaps on the same arm. Outside this restriction in range of motion, the flaps are unrestrained and are actuated only by the flow past them, giving the turbine self-start capability. Figure 1 shows a schematic diagram of the CarBine mechanism, where the body is rotating at an angular velocity ω (rad/s) in a flow with a velocity U_∞ (m/s), along with a 3D rendering of the device. This design aims to utilise the strengths of vertical axis designs for a varied set of uses, without compromising aspects such as the geometrically simple components, with deployment scenarios of tidal regions and as small hydrokinetic installations.

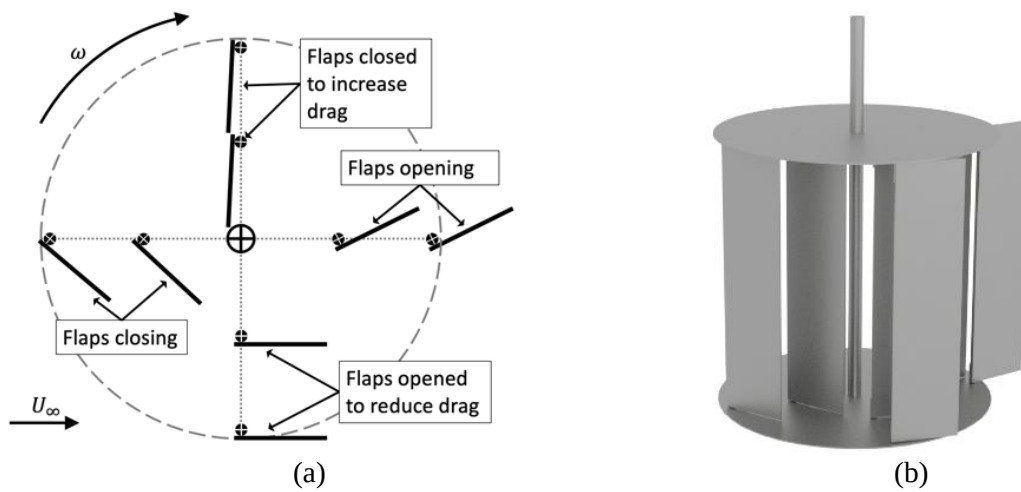


Fig. 2 Cross-sectional diagram (a), and 3D rendering (b) of CarBine

Laboratory tests already carried out at Cardiff University, were to be used for validation and calibration of the numerical modelling configuration, before being expanded in scope to explore alternative configurations and conditions. These experimental tests were run in a flume with a rectangular cross-section 1.2 m wide, 1.0 m high, and 17.0 m long with the capability to operate between 10 and 1000 l/s as described by Rauen et al. [14], shown in Figure 3.

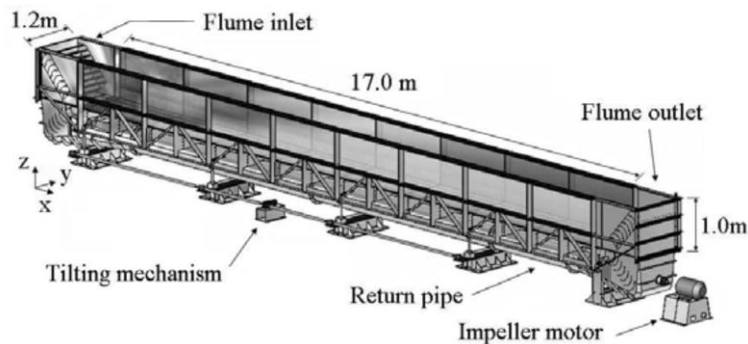


Fig. 3 Cardiff University NERC flume 3D perspective (adapted from [14])

The inflow velocities used in the flume experiments were 0.51, 0.64, 0.77 and 1.08 m/s. An Acoustic Doppler Velocimeter (ADV) was used to measure the flow velocity in the flume at a number of positions across the section, shown in Fig. 4. These measurements were taken with the flume operating without the turbine in position, and the flume

configurations settings stored and thus re-applied with the turbine model installed. An average free-stream velocity was calculated using the combined profile, calculated over the region which the turbine occupies using the measured vertical, streamwise, and spanwise velocities. The vertical velocity profile shown in Fig. 4Error! Reference source not found.b is highly logarithmic, and the average flow over the turbine may vary with changes in the three-dimensional flow profile and relative position of the device in the water column. For simplicity, the flume average was used as the input velocity for these tests, with full calculations available in Ranabhat [15].

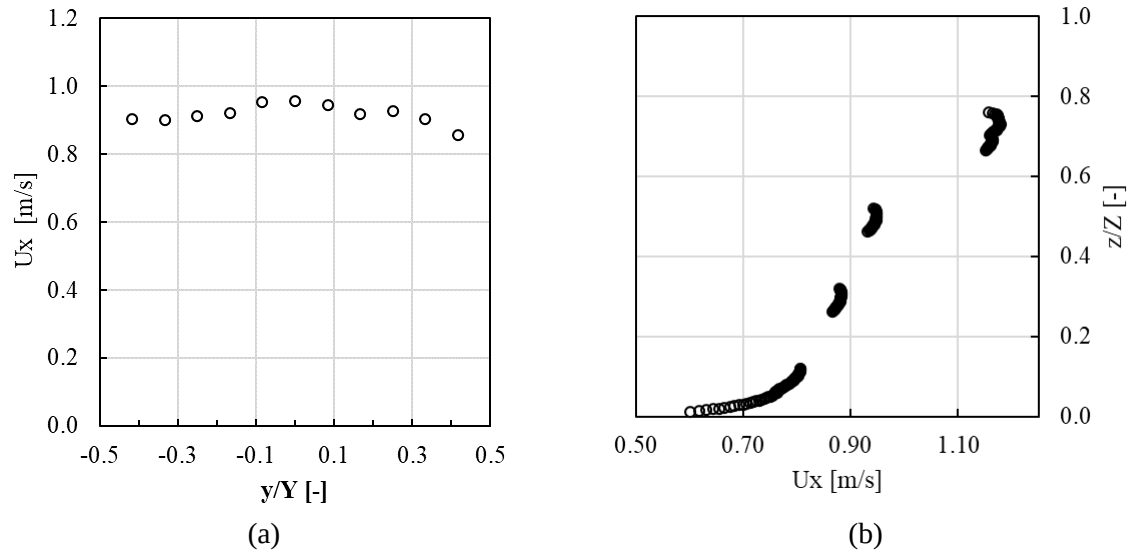


Fig. 4 Measured flume flow velocities (a) across the flume, and (b) in the vertical direction

One of the disadvantages of the CarBine design is the need for multiple moving parts; the ability of the turbine to operate whilst damaged and without a component is beneficial in scenarios where such a device would be the primary source of power. To provide some insight into how the base design would operate if damaged, and to apply the model to a scenario that provides additional insight, two tests were carried out. Each case involved removing a single flap and replacing it with a simple pin (as in the prototype the flaps are rotationally restrained by the axle connections) simulating the loss through damage of a single flap: loss of the inner flap (Case 1) and outer flap (Case 2).

Physical modelling and field testing are important components in the development of a tidal turbine. However, they can be expensive, time-consuming, and scaling assumptions are typically required where space is not available for prototype scale models [16]. Numerical models allow the designer to make changes and investigate performance changes across various scenarios without the limitations of scaling or physical lab space. There are many approaches available in the field of Computational Fluid Dynamics (CFD) for the analysis of hydro turbine behaviour. Židonis et al [17] compared a variety of Eulerian (mesh-based) and Lagrangian (mesh-free) methods/software used in the design of a Pelton turbines using both 2D and 3D modelling tools. Ansys CFX and Ansys Fluent were found to be most commonly used, with high accuracy and precision paired with a high computational cost, and noted that Lagrangian methods can be useful in the earlier design stages of such devices.

Allmark et al. [18] demonstrated the process for testing a conventional style horizontal axis tidal turbine using both physical and numerical models in a numerical study (carried out using Ansys CFX) where the turbine was represented in a Moving Reference Frame (MRF); a method whereby the turbine and adjacent space were modelled in a cylindrical region, that rotated at a prescribed angular velocity, determined from lab study or empirical analysis [19]. This method however required the rotational velocity of the turbine at a given flow speed and torque output to be identified in advance, and the turbine itself to not change shape within the MRF. To accurately represent the CarBine design using traditional methods would require nested MRFs, whereby each flap is in a reference frame of its own, within the MRF of the body itself, and this would still require that the individual flaps are able to move into and out of the regions deemed belonging to other flaps, as visible in Figure 1, the opening inner flap is in the space occupied by the closing outer flap. This nesting and cross-availability of

systems would require a non-trivial model construction (potentially achievable using approaches such as dynamic meshing), and as such would be less suitable for explorative early-stage design modifications. Large Eddy Simulation with an Immersed Boundary (LES-IB) has been used to predict the performance of a vertical axis Darrieus turbine under both laminar and turbulent flow conditions by Ouro and Stoesser [20] using both 2D and quasi-3D simulations (3D domain not considering end effects) wherein the Lagrangian turbine rotates at a prescribed constant rotational velocity within an Eulerian fluid domain, again requiring a known motion of the body prior to modelling. Salehi et al. [21] used a flexible mesh to adjust the guide vanes of a Francis turbine over the process of a turbine shut-down using the open-source CFD solver OpenFOAM.

The Hunter turbine (patent no. GB9524439.8), shares some conceptual elements with CarBine [22, 23]. Yang and Lawn [22] ascertained geometric positions for the Hunter turbine at given body angles and used these positions to determine the forces across the blades and as such the turbine as a whole using a 2D Reynolds Averaged Navier Stokes (RANS) CFD model (using the commercial FINE™/Turbo code). Using these positions, Hashemi et al. [23] developed equations to describe the flap positions based on the body angle and implemented these into a 2D model using Autodesk CFD where the mesh is deformed to form the predefined motion of the turbine over time.

The CarBine design has a key complexity in that the turbine itself is composed of a set of linked but free-moving components. The torque produced by the design in any position is not simply a factor of the fluid moving past it, as the positions of the flaps are constantly changing under the influence of the flow. A method to assess the performance of the design without a requirement for prior knowledge of the minutiae of operation was deemed necessary, with the classical options each proving to lack some elements. Additionally, a desire to mitigate costs both computationally and financially, as these can be seen as entry barriers to progression, was seen as important.

Smoothed particle hydrodynamics (SPH), originally developed in the field of astrophysics (simultaneously) by Gingold and Monaghan [24], and Lucy [25], is a Lagrangian meshless method that solves the Navier-Stokes equations over a domain containing particles effectively representing units of fluid. These particles enact forces upon each other based on a kernel function and can be held in fixed relations to represent solid bodies, free to move under the influence of the flows around them, with material parameters such as density being related to the particles themselves. The use of SPH has expanded to a diverse range of engineering applications including fluid-structure interaction [26], wave-structure interaction (both in 2D [27–29] and 3D [30]), multi-phase flow problems [31–33], and floating body analysis [34–36]. The potential of SPH to represent situations with a complexity that is prohibitive to traditional mesh-based numerical modelling could complement mesh-based methods and allow for better insights into an array of engineering situations [37]. SPH was identified as a method that could provide insights into the design and operation of the turbine, having been used by Marongiu et al. [38, 39] to simulate the complex flows and rotating geometry within a Pelton turbine using a 3D hybrid Arbitrary Lagrange Euler (SPH-ALE) method, with Rentschler et al. [40] confirming the robustness of this approach. The complexities of representing this turbine concept numerically are well mitigated by the SPH method, and with the solid body interaction capability of CHRONO [41] – the DualSPHysics software was found to be an ideal candidate for this work.

Both 2D and 3D models have been used to analyse vertical-axis turbines at different stages of the design and to achieve different objectives [16, 22, 42]. Roy and Saha [43] stated that the use of a 2D assumption in a vertical axis turbine is suitable when the height is greater than or equal to the diameter of the device. Ouro and Stoesser [20] noted that 2D simulations generally overestimate performance when compared to their 3D counterparts, highlighting the importance of detailed 3D numerical or experimental modelling to confirm what a 2D study has ascertained. To provide initial proof of concept, a 2D representation was used at this stage. This was done by taking a plan section of the device. For early-stage device assessment, the value of a reduced computational load is valuable, saving time and cost to enable explorative design development, in SPH performance is strongly influenced by the number of particle interactions, with each particle interacting with typically 40 neighbours in 2D and 250 in 3D kernel formulations [44]. As the simulations used ~2.5 million particles, every additional layer of particles into the third dimension would add the same number of particles at generation and increase the runtime to a greater degree. Thus, the increased degree of insight gained from a 3D representation was deemed to be dominated by the cost and potential complications incurred at this stage.

The primary objectives of this paper are to assess the potential of SPH to quickly and simply model a vertical axis turbine – without prescribing a rotational motion, and thus to use the SPH model to provide new insight into the design of this turbine such as hydrodynamic self start. To achieve this, a 2D SPH model (described in Section 2) was calibrated, validated, and tested against a range of base flow conditions measured in the laboratory. The validated model was then applied to simulate the turbine, assessing the resilience of the base design to damage (a hitherto untested scenario). Section 2 describes the modelling approach, the results of the modelling are presented and discussed in Section 3, and conclusions are brought forth in Section 4.

2. METHOD

The modelling was carried out using DualSPHysics, as described by Domínguez et al. [44]. This is an open-source SPH package utilising the Nvidia CUDA programming toolkit to increase computational speed through the use of the Graphical Processing Unit (GPU) [45]. DualSPHysics allows coupling with the multi-body solver, Project Chrono [41], which can be used to solve interactions and provide connections between floating and fixed solid bodies within the fluid domain – as described by Martínez-Estévez et al. [46], which is key to appropriate representation of CarBine. The DualSPHysics model was predominantly run on the Supercomputing Wales HPC, Hawk, using Nvidia Tesla P100 and Nvidia Tesla V100 GPU-enabled computational nodes, with performance comparison models run on a desktop workstation with an Nvidia Quadro P2000 GPU. In the following, an overview of the SPH method used is described.

2.1 SMOOTHED PARTICLE HYDRODYNAMICS

To represent the behaviour of an incompressible fluid the SPH model uses the Navier-Stokes governing equations for a weakly-compressible fluid in the Lagrangian forms of the continuity (1) and momentum (2) equations:

$$\frac{d\rho}{dt} = -\rho \nabla \cdot \mathbf{v} \quad (1)$$

$$\frac{d\mathbf{v}}{dt} = -\frac{1}{\rho} \nabla P + \Gamma + \mathbf{f} \quad (2)$$

Where ρ is the fluid density, t is time, $\mathbf{v}$ is the velocity (bold lettering hence indicates vector notation), P is the pressure, Γ the dissipative terms, and $\mathbf{f}$ is an external source acceleration vector (typically gravity). The mathematical principals of SPH are based on integral approximation of a continuous function at a position $\mathbf{r}$ over a domain Ω , here denoted by $F(\mathbf{r})$ (3).

$$F(\mathbf{r}) = \int_{\Omega} F(\mathbf{r}') W(\mathbf{r} - \mathbf{r}', h) d\mathbf{r}' \quad (3)$$

Where the approximation of the function at a position $\mathbf{r}$ is calculated over a compact support region of radius $2h$ (where h is defined as the smoothing length) using a so-called smoothing kernel function W to weight the contribution of the surrounding region. Multiple kernel functions are available in DualSPHysics; the Quintic Wendland kernel (4) was used here (α_D ensures that $\int W(|\mathbf{r} - \mathbf{r}'|) d\mathbf{r}' = 1$, and α_D is equal to $7/4\pi h^2$ for 2D implementations), where $0 \leq q \leq 2$ and $q = \frac{|\mathbf{r} - \mathbf{r}'|}{h}$:

$$W(q) = \alpha_D \left(1 - \frac{q^4}{2}\right) (2q + 1) \quad (4)$$

By considering the domain to be a collection of computational nodes or particles, Equation (3) can be evaluated for a particle a , located at position $\mathbf{r}_a$ with a set of neighbour particles b with positions $\mathbf{r}_b$, masses m_b , and densities ρ_b through the following

$$F(\mathbf{r}_a) \approx \sum_b F(\mathbf{r}_b) W(\mathbf{r}_a - \mathbf{r}_b, h) \frac{m_b}{\rho_b} \quad (5)$$

In Equation (2), the momentum source term $\mathbf{f}$ is hence taken as gravity $\mathbf{g}$, the dissipative term Γ was handled in DualSPHysics by the Artificial Viscosity scheme described by Monaghan [47], is added to the pressures terms whereby the particle has velocity $\mathbf{v}_a$, P_a and P_b are the pressures at a and b respectively, resulting in the momentum term of the Navier-Stokes equations in discrete Lagrangian form (6). Π_{ab} is the viscosity term (7), formed by the relative positions and velocities of the particles, where $\bar{c}_{ab}$ is the average speed of sound across a and b , $\bar{\rho}_{ab}$ the average density, ∇_a the gradient operator, W_{ab} is the value of the kernel function weighting of b on a and finally, α is a dimensionless tuning coefficient known as the artificial viscosity that needs ascertaining based on the model aims. This viscosity term thus produces a repulsive force on two particles approaching, and an attractive force on particles moving apart, based on their relative positions $\mathbf{r}_{ab}$, and velocities $\mathbf{v}_{ab}$.

$$\frac{d\mathbf{v}_a}{dt} = - \sum_b m_b \left(\frac{P_a + P_b}{\rho_a \cdot \rho_b} + \Pi_{ab} \right) \nabla_a W_{ab} + \mathbf{g} \quad (6)$$

$$\Pi_{ab} = \begin{cases} -\alpha \frac{\bar{c}_{ab} \mu_{ab}}{\bar{\rho}_{ab}}, & \mathbf{v}_{ab} \cdot \mathbf{r}_{ab} < 0 \\ 0, & \mathbf{v}_{ab} \cdot \mathbf{r}_{ab} > 0 \end{cases} \quad (7)$$

$$\mathbf{r}_{ab} = \mathbf{r}_a - \mathbf{r}_b, \text{ and } \mathbf{v}_{ab} = \mathbf{v}_a - \mathbf{v}_b$$

$$\mu_{ab} = h \frac{\mathbf{v}_{ab} \cdot \mathbf{r}_{ab}}{\mathbf{r}_{ab}^2 + 0.01h^2}$$

In complement to (6), the conservation of mass in SPH form for a weakly-compressible fluid is given by (8). Here the compressibility of the fluid is governed by the equation of state for the fluid (9) based on Monaghan [47]. This is developed from the relationship between the pressure P , the polytropic constant γ , the reference density ρ_0 , the particle and reference speeds of sound c and c_0 respectively, and B is a dimensionless parameter to mitigate density fluctuations [47].

$$\frac{d\rho_a}{dt} = -\rho_a \sum_b \frac{m_b}{\rho_b} \mathbf{v}_{ab} \cdot \nabla_a W_{ab} \quad (8)$$

$$P = B \left[\left(\frac{\rho}{\rho_0} \right)^\gamma - 1 \right] \quad (9)$$

$$\gamma = 7, B = c_0^2 \rho_0 / \gamma, \rho_0 = 1000 \text{ kg/m}^3, c_0 = c(\rho_0) = \sqrt{\partial P / \partial \rho} \Big|_{\rho_0}$$

To reduce density (and therefore pressure) fluctuations, the density diffusion term (DDT) is added to (8) using the DDT Ψ_{ab} as proposed by Fournakos et al. [48], where the coefficient δ_Φ of 0.1 is recommended for most applications, leading to the conservation mass in the form (10).

$$\frac{d\rho_a}{dt} = -\rho_a \sum_b \frac{m_b}{\rho_b} \mathbf{v}_{ab} \cdot \nabla_a W_{ab} + \delta_\Phi h c_0 \sum_b \Psi_{ab} \cdot \nabla_a W_{ab} \frac{m_b}{\rho_b} \quad (10)$$

To mitigate the excess propagation of voids within the domain, Lind et al. [49] applied Fick's first law of diffusion to control shifting, the particle flux $\mathbf{J}$ being proportional to the concentration gradient ∇C by a degree of the Fickian diffusion coefficient D_F (11).

$$\mathbf{J} = -D_F \nabla C \quad (11)$$

The discrete shifting distance $\delta \mathbf{r}_{a_s}$ for a particle is thus given by (12) using the kernel gradient ∇C_a , and the shifting diffusivity D_s – which in turn is a function of the absolute particle velocity magnitude $\|\mathbf{u}\|_i$, and timestep dt (13) as proposed by Skillen et al. [50] A is a new parameter that controls the degree of shifting and maintains proportionality.

$$\Delta \mathbf{r}_{a_s} = -D_s \nabla C_a \quad (12)$$

$$D_s = Ah\|\mathbf{u}\|_i dt \quad (13)$$

In this modelling approach, the turbine is represented as a fluid-driven object, composed of multiple rigid bodies. As such, the forces f felt by each boundary (turbine) particle k from the surrounding fluid particles a are given by f_k (14). Thus, for a body with mass M , composed of a set of boundary particles (BPs) the Newtonian motion V is derived from the sum of the contributions of the particle set (15) where each particle has mass m_k and contributes force f_k . The inertia I , rotational velocity Ω , and centre of mass $\mathbf{R}_0$ are related under the same principal per (16).

$$f_k = \sum_{a \in WPs} f_{ka} \quad (14)$$

$$M \frac{dV}{dt} = \sum_{k \in BPs} m_k f_k \quad (15)$$

$$I \frac{d\Omega}{dt} = \sum_{k \in BPs} m_k (\mathbf{r}_k - \mathbf{R}_0) \times f_k \quad (16)$$

The motion of the whole body is thus determined by integrating (15) and (16), and the motion of each boundary particle in the body is therefore calculated (17). In the case used here, some external restraints, along with the solid body interactions, are handled externally by the solid body solver, Project Chrono [41], $\frac{dV}{dt}$, $\frac{d\Omega}{dt}$ and Δt are passed from DualSPHysics to Chrono, which calculates the solid body interactions and returns V , Ω , and $\mathbf{R}_0$.

$$\mathbf{v}_k = \mathbf{V} + \Omega \times (\mathbf{r}_k - \mathbf{R}_0) \quad (17)$$

The Chrono library [41] includes several connection methods for bodies within the model, which was used here to appropriately connect the components of the turbine. The body was connected to the domain by a hinge which acted as an axle. The flaps were then connected to the body, and so hinged about their individual axles, and when in the closed position then transferred the forces through them to the body and thus turn the turbine as a whole. The configuration is shown in Figure 5, note that hinges can be placed beyond the physical position of a component when using Chrono, allowing bodies to be connected about a notional point – and floating to floating connections are maintained relative to the forces on the bodies and the centroidal positions of the components.

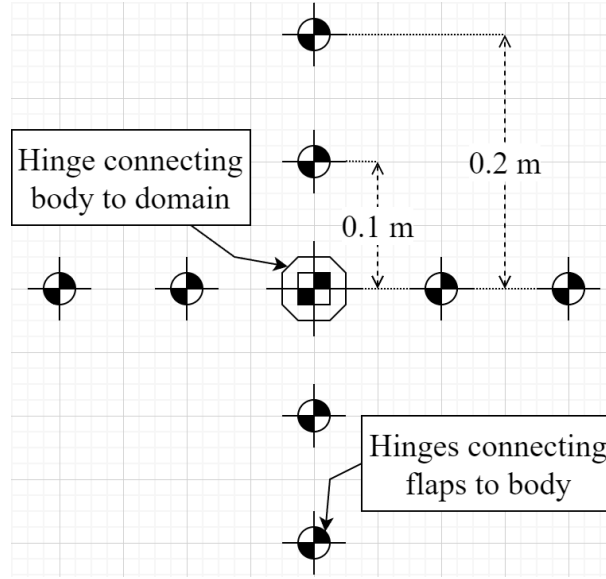


Fig. 5 Chrono configuration diagram, showing relative positions of CHRONO hinges used to represent turbine connections.

2.2 MODEL SETUP

The SPH model was set up as a 2.0 m long section (5 turbine diameters) of the flume which included the turbine as shown in Fig. 6. The flume inflow boundaries were simulated utilising the procedure presented by Tafuni et al. [51]; the upstream end was configured as an inflow buffer zone with a constant uniform velocity of U_∞ using 8 layers of fictitious particles outside the domain with a prescribed velocity from which particles were extruded to maintain the flow. The outflow buffer used extrapolated fluid properties onto 8 layers of fictitious nodes which extracted their properties from ghost nodes within the domain. The flume side walls were represented using an impermeable boundary using the modified Dynamic Boundary Condition (mDBC) described by English et al. [52] wherein ghost nodes were mirrored across the solid boundary to satisfy the continuity equation, and a no-slip condition was applied by fixing the boundary particle velocity to be zero. The boundary condition used for the turbine was DBC [53] due to mDBC not being compatible with Chrono objects in the version of DualSPHysics used (v5.0), as well as being more repulsive and thus mitigating interaction over slender elements. The impact of the position of the downstream boundary on turbine operation was assessed by comparing the motion of the CarBine with the boundary placed at a range of distances as discussed in Section 2.5. Hydrodynamically driven net motion of the device (the cumulative turning of the central axle – which would be connected to a power take-off in an operational prototype) was chosen as a default calibration parameter as a key novelty of the approach was the absence of prescribed motion.

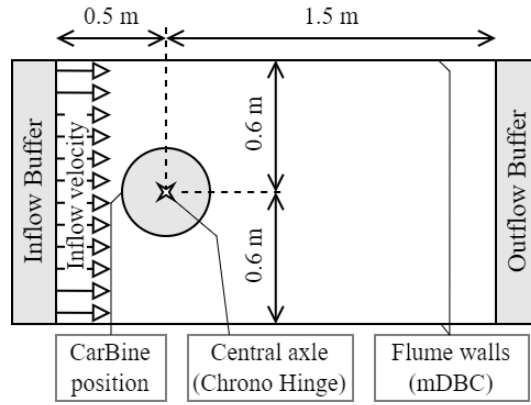


Fig. 6 Schematic of numerical model setup

The SPH scheme was configured using the Symplectic predictor-corrector time integration scheme, whereby the timestep was dynamically determined based on the Courant-Freidrichs-Lewy (CFL) condition [54] limited to a CFL value of 0.2. The quintic Wendland kernel function [55] with a smoothing_length h of 2.25 times dp where dp (m) was the initial particle spacing, and the Density Diffusion Term of Fourtakas et al. (10) [48]. However, with the gravitational forces set to zero in all directions – the body forces are therefore also eliminated and this formulation reduces to that of Molteni and Colagrossi [56]. Due to the interactions of the flaps, the model was highly prone to the formation of voids due to the opening and closing of the flaps under the pressure of the flow. To mitigate this issue, and preserve the consistency of the particle spacing, the particle shifting algorithm of Skillen et al. [50] (13) was used with the ‘A’ shifting coefficient needing to be at the extreme value of -50 (default -2, with values of -20 used in some 2D example/benchmark cases). Shifting magnitudes of -1, -5, -25, -50 and -100 were tested, smaller magnitudes led to the development of unphysical voids within the model domains at areas of low pressure, as shown in Figure 7. Higher shifting magnitudes led to increased computational time (a 20 % increase from -1 to -100) and so the smallest stable value (-50) was used. In 3D cases this effect could also be mitigated using Tensile Instability Control (TIC) as described in Sun et al. [57].

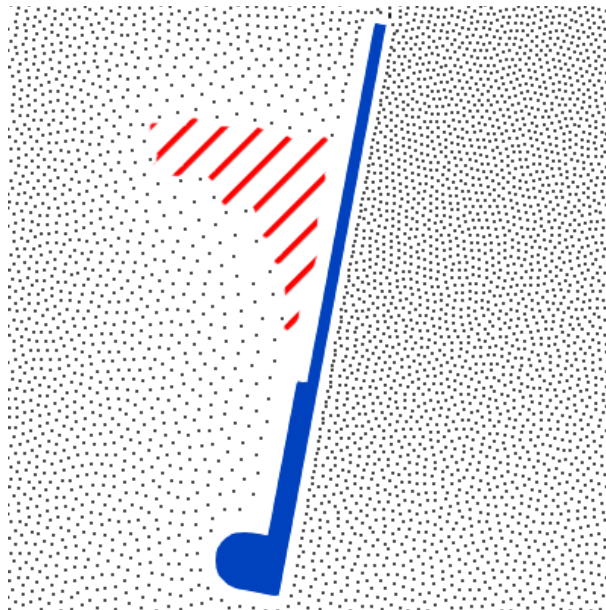


Fig. 7 Void formed during test at shifting value equal to -25, void shown in red hashing, with particles as black points and blade rendered as blue solid.

The free surface detection parameter was set to 0.0 (no free-surface) as no free-surface interaction was expected within the domain – further mitigating the formation of voids. The artificial viscosity treatment described by Monaghan [47] was used to stabilise the model and treat the effects of dissipation. The Sub-Particle Scale (SPS) turbulence scheme was first introduced by Gotoh et al. [58] for Moving Particle Semi-implicit (MPS) modelling, before being adapted to incompressible SPH by Gotoh et al. [59] and then weakly compressible SPH by Dalrymple and Rogers [60]. The Laminar + SPS scheme in DualSPHysics was tested at a variety of conditions but the interaction of low-pressure regions and voids with the outlet boundary when using this formulation resulted in large instabilities in the domain – and so the more robust artificial viscosity was used. The dimensionless α (7) tuning parameter has been found to converge varyingly depending on the problem in question [61]. In this study, a dimensionless tuning parameter value of $1e^{-3}$ to $1e^{-6}$ was found to produce consistent results when compared to flume measured turbine rotation, with $1e^{-5}$ producing the most stable operation.

The artificial viscosity formulation provides no inherent turbulence model, whereas the alternative (Laminar + SPS) formulation does [44]. Hence, an alternative approach was required. Direct numerical simulation of turbulent effects (using a resolution so fine that turbulent effects are calculated) has been carried out using SPH both in the case of imposed turbulent conditions being allowed to dissipate [62, 63] in 2D simulations, and in the study of fluid-mixing devices in 2D representations where an encouraging agreement to prior studies was found [64, 65]. To achieve this, a high-resolution initial particle spacing was used in the studies, varying from 0.375 to 6.667 mm in [64] and [63], respectively. The selected values of artificial viscosity were chosen to best match the numerical Reynolds number to that of the experiment [66]. The particle spacing and smoothing length chosen to be small enough to best approximate direct numerical simulation of the turbulent effects and while still allowing for simulations to be run in an acceptable time.

2.3 CALIBRATION

For the model studied here, a high resolution was required to reach convergence of results as demonstrated in Fig. 8, using the L2 error norm metric per Equation (18). For this study the net displacement of the turbine ϕ (radians) outside of an initial start-up phase was considered for use in assessing the accuracy of the model, as predicting the rotation of the turbine is one of the main focus points of the study. Quinlan et al. [67] found that the error in discretizing a fluid domain into a particle field depends on the particle spacing itself (dp) and the ratio of dp and h (dp/h), although this second error may dominate when the physical size of h drops below a critical value. As such it is necessary to ascertain convergence to both dp and dp/h , for the specific case being assessed. Due to the slender nature of the flap blades (2 mm at the narrowest point), a particle resolution of less than 2 mm was deemed necessary, with dp of 0.5, 1, 1.5, and 2 mm tested at h/dp ratios of 1.5 to 2.25 as suggested by Quinlan et al. [67]. At 0.5 mm spacing (dp), the error arising from the particle spacing to smoothing length ratio (dp/h) was the dominant error, resulting in lower accuracy than the coarser (and computationally cheaper) spacings, and as such the 0.5 mm spacing was excluded from further assessment.

$$L2(\phi) = \frac{1}{N} \sqrt{\sum_{i=1}^N \frac{(\phi_{Flume} - \phi_{SPH})^2}{\phi_{Flume}^2}} \quad (18)$$

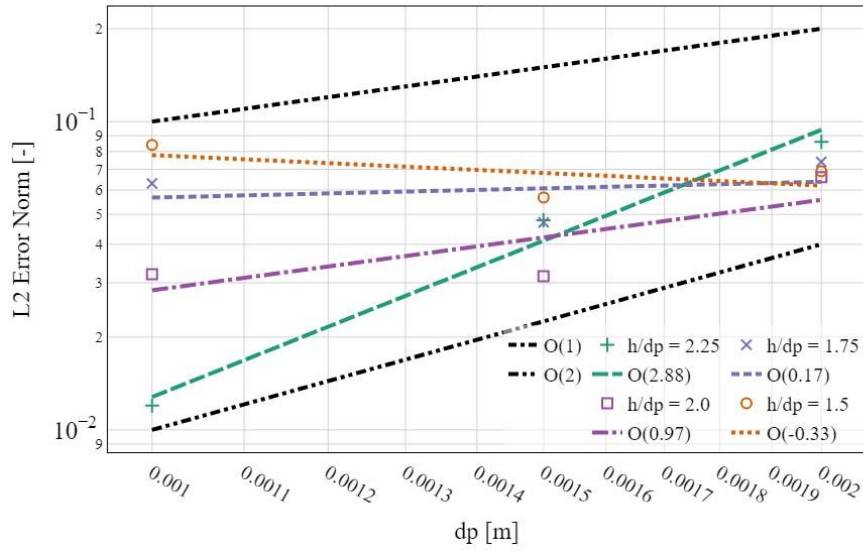


Fig. 8 Convergence of model particle spacing (dp) at varied h/dp ratios

At h/dp of 2.25, the models approached third-order convergence with the convergence order decreasing along with h/dp – in line with what would be expected per Quinlan et al. [67]. This extreme rate of convergence may stem from the relatively coarse assessment parameter in net turbine rotation. Ideally an analytical solution should be used to assess model performance. In the absence of an analytical solution, measurements were used, which may contain errors. Notably, the slenderness of the flap ends was again critical when testing h/dp values in excess of this value, as too high of a smoothing length allowed disruptive levels of particle interaction across the blades, causing errors during operation. It is likely that for a turbine design with bulkier components, increasing this parameter may yield further performance improvements. For each given dp , the accuracy can be seen to improve with increasing h in Fig. 9, with the order of convergence being in excess of second-order (shown as $O(2)$ in Fig. 9) for the 1 mm spacing. Based on these tests, 1 mm dp , with a h/dp of 2.25 were utilised in the remainder of the modelling.

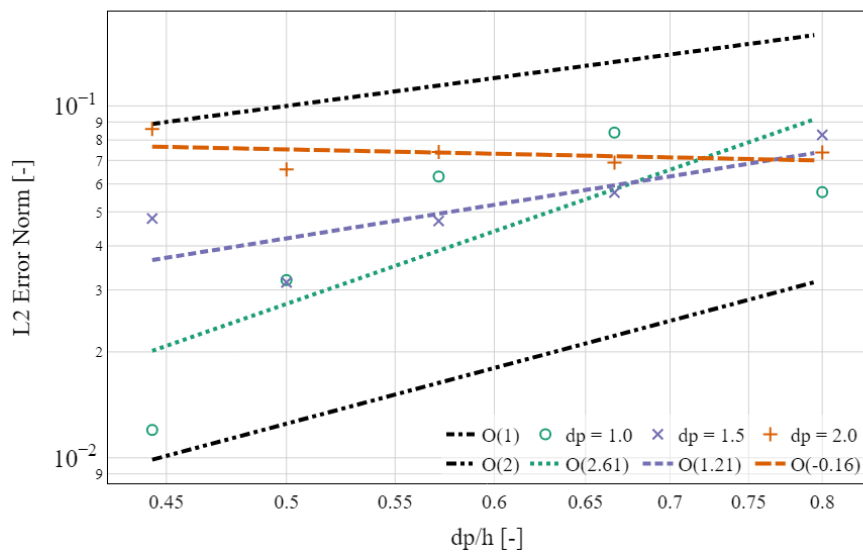


Fig. 9 Convergence of model to dp/h at varied particle spacings (dp in mm)

The initial particle distribution for DualSPHysics was computed using the GenCase tool. An initial particle spacing about the central axle is shown in Fig. 10 (full design descriptions of various prototype configurations of CarBine can be found in Harries et al. [13] and Ranabhat [15]). This tool partitions the domain into a regular cartesian grid with spacing based on the parameter dp . For certain shapes, the ‘freedraw’ option can be enabled to allow the particles to better match the shape. For example, in a circle, as used in the central axle, the regular spacing by default could not provide a good match and therefore freedraw was used to improve the representation.

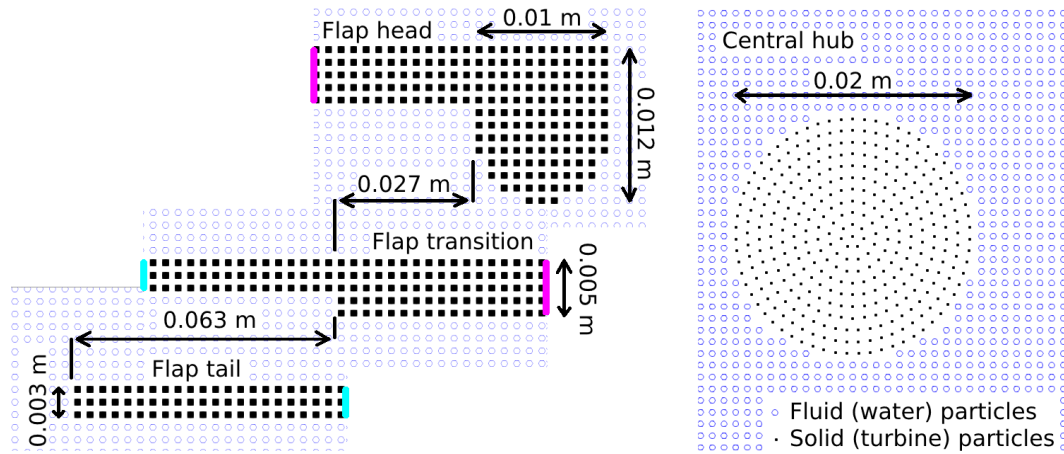


Fig. 10 Initial particle spacing around the central axle, with design details shown for (truncated) flap, particles generated at $dp = 1$ mm

2.4 MODEL APPLICATION – RESILIENCE

Once a hydro-kinetic device is installed, it is likely that it would experience a variety of events likely to cause damage. Kirke [7] highlighted damages to the turbine from floating bodies, such as logs, as one of the main problems facing small turbines. A device installed to produce electricity in a remote location, where access to the turbine, availability of parts and alternative sources of electricity may be limited, must be able to operate to some degree whilst damaged. This is particularly important as damaging conditions may occur at times of greater stress such as floods or storms when loss of power would be additionally detrimental. To address the effectiveness of the turbine under some post-damage scenarios, a set of tests were outlined to test the performance of the device with a single flap removed intentionally to represent a broken flap as a result of floating debris, and replaced with a simple pin. This is consistent with the experimental prototype where the hinge connection points to the body provided that support. The testing of an asymmetrical device configuration operating in an unknown condition provides an opportunity to demonstrate the ability of the SPH method to perform analysis that would be very challenging in mesh-based methods. The tests sought to provide initial insight into how the device will behave post breakage, which flaps were most critical to the operation of the device, and if a given flap is lost, could the remaining flaps be relocated in some way to provide the maximum energy whilst in the damaged state, as resilience to damage was identified as a key design motivator [7].

2.5 BOUNDARY LOCATION

Though the wake characteristics of this device are not the focus of the study, the position of the downstream boundary and its interaction with the wake do have the potential to impact the motion of the turbine during the test. To ensure the boundary was not having an influence on the test, a set of tests were run whereby the position of the boundary was varied from 2 – 5 m from the inlet, in 0.5m increments, with the turbine positioned 0.5 m from the inlet. Also, an extra run was carried out with an increased total length of 7.5m. It was found that the position of the downstream boundary had a minimal effect on the displacement of the device (mean L2 error of 0.0209). To reduce the computational cost,

the smallest domain (a) was utilised in the remainder of the tests. A rendering of the boundary position tests at the end of the runtime is shown in Figure 10, with the angular displacement over time shown in Figure 11.

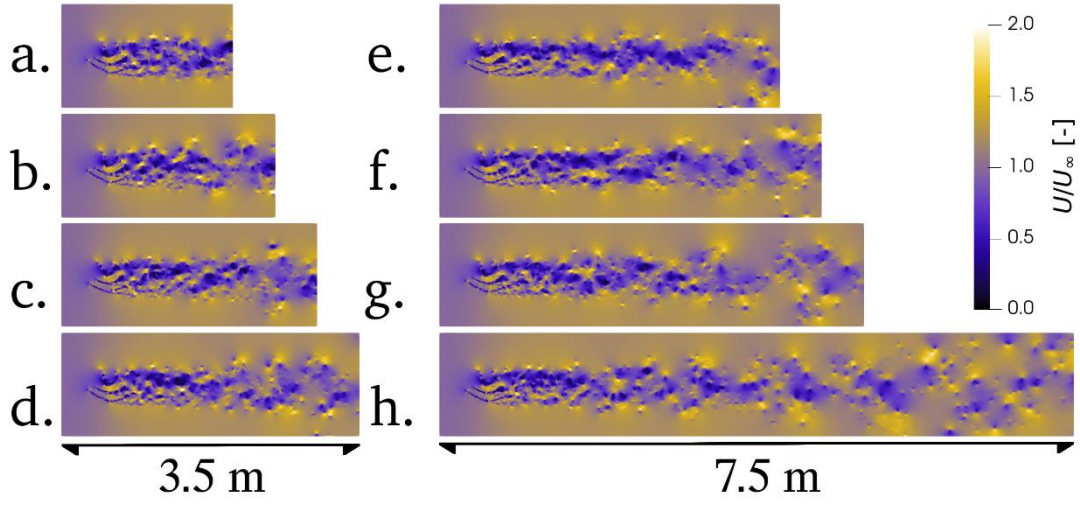


Fig. 11 Flow velocity profiles normalised to inlet velocity at $t = 10$ s using multiple domain lengths, at $U_\infty = 0.9$ m/s

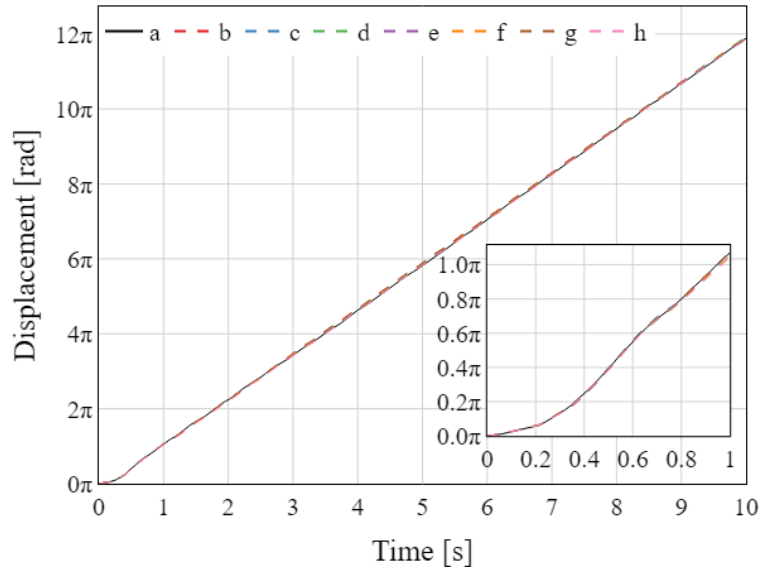


Fig. 12 Turbine body displacements using multiple domain lengths start-up period detail inset, at $U_\infty = 0.9$ m/s

3. RESULTS AND DISCUSSION

The base model condition, with U_∞ of 0.9 m/s (which was found to be a point of optimal performance in flume testing [15]), dp of 1 mm, and h/dp of 2.25 was computed on three platforms to provide a variety of representative performance rates. The first was an Nvidia Quadro P2000 in a desktop workstation, the second and third being GPU-enabled nodes on the Supercomputing Wales HPC – Hawk. These are equipped with either Nvidia Tesla P100 (Pascal architecture), or Tesla V100 (Volta architecture) processing units each with 16 GB of memory. This model configuration has on

average – a maximum of ~ 2.5 million particles, and ~ 193 Tera Particle Interactions (TPIs). The Quadro P2000 desktop completed the simulation with an average of 10102 seconds of runtime per physical second (s/s), the P100 operated at 2870 (s/s), and the V100 was the fastest at 1655 (s/s). The Tesla V100 was used in the performance analysis carried out in [44] and found to be in excess of 200 times faster than a 12-core CPU – highlighting the advantage of carrying out the calculations on a GPU-enabled device.

3.1 BASE MODEL

To understand the degree to which the modelled turbine was able to represent the physical prototype, the base four-arm model was placed in a flume with an incident velocity U_∞ of 0.9 m/s across the flume from an assumed start-up position. The flume model had a mean angular velocity of 3.87 rad/s over the tests, with the SPH model having a mean velocity of 3.825 rad/s excluding a 1-second warm-up – a 1.2% difference in average rotational velocity. This relationship can be seen in Fig. 13 (with the averaged flume rotation rate plotted). From this timeseries, the ability of the SPH model to capture high-frequency effects is demonstrated, with the flume velocity being ascertained from a 1 Hz sampling rate, whereas the SPH model printed results at a 50 Hz rate demonstrating some high-frequency periodic behaviour with oscillations of approximately 5 Hz. This plot also shows the peak rotational speed experienced during the hydrodynamically driven self start, an effect which could not be captured in previous studies of the turbine.

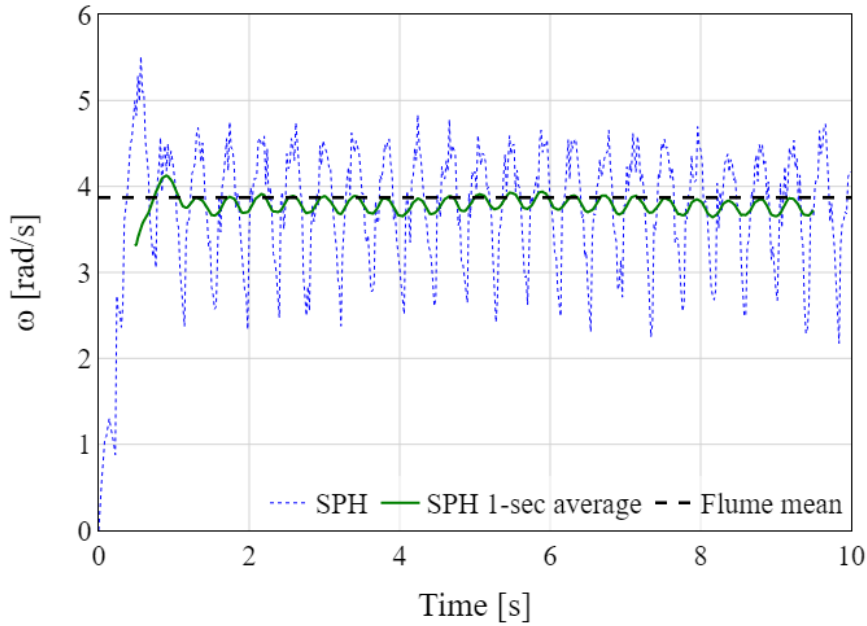


Fig. 13 Flume and SPH model angular velocity at U_∞ of 0.9 m/s

In assessing the rotational speed of a tidal stream turbine across a variety of free stream velocities, the dimensionless tip speed ratio λ (-) is used, which is calculated as follows [13]:

$$\lambda = \frac{r_T \bar{\omega}}{\bar{U}_\infty} \quad (19)$$

where λ is the Tip Speed Ratio (TSR), r_T (m) represents turbine radius (here the distance from the turbine centre to the outer hinge pin), $\bar{\omega}$ (rad/s) is the mean angular rotation of the turbine, and $\bar{U}_\infty$ (m/s) is the mean free stream velocity. To ascertain how the speed of the turbine varies as it rotates at a higher frequency, the tip speed ratio was calculated at every timestep and plotted against the body angle for the base condition, shown in Fig. 14. Turbine motion is clockwise in this case and throughout this study. The start-up motion of the turbine is visible in the lower tip speed ratio point cluster around the 0 radian angle, after which the initial rotation of the turbine is shown by the set of points at higher TSR

than the generally consistent in-operation motion. Once under motion, the turbine exhibits four-way symmetry as can be expected from a four-armed device, with a rapid acceleration as the blades fall into position followed by a mild acceleration, and finally a slowing once the turbine passes the optimal point. The rapid start-up phases shown indicate a portion of the behaviour that could require additional study of the component interactions as this sudden acceleration due to the flaps impacting the body is indicative of a peak force occurring and thus the greatest potential for damage.

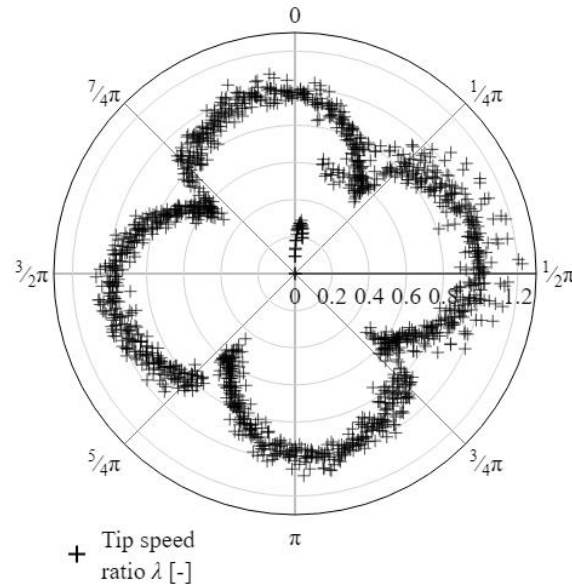


Fig. 14 Tip speed ratio at body pitch angles for the base condition ($U_{\infty} = 0.9$ m/s)

Renderings of the particle velocities around the device at the end of a model run normalised to the incident upstream velocity is presented for the domain in Fig. 15 with a detail showing the spacing of individual particles at initial generation in Fig. 10. Here, along with the vorticity field (Fig. 16), the turbine produced an unsteady wake structure that was absorbed by the downstream boundary. The turbulence, wake effects, and how they could, for example, influence a potential array structure are topics upon which further research could be conducted but was beyond the scope of this study. Fig. 17 shows the pressure distribution across the domain calculated per Equation (9) for each particle, demonstrating how an increased pressure was developed upstream of the driving blades, against the heads of the returning blades.

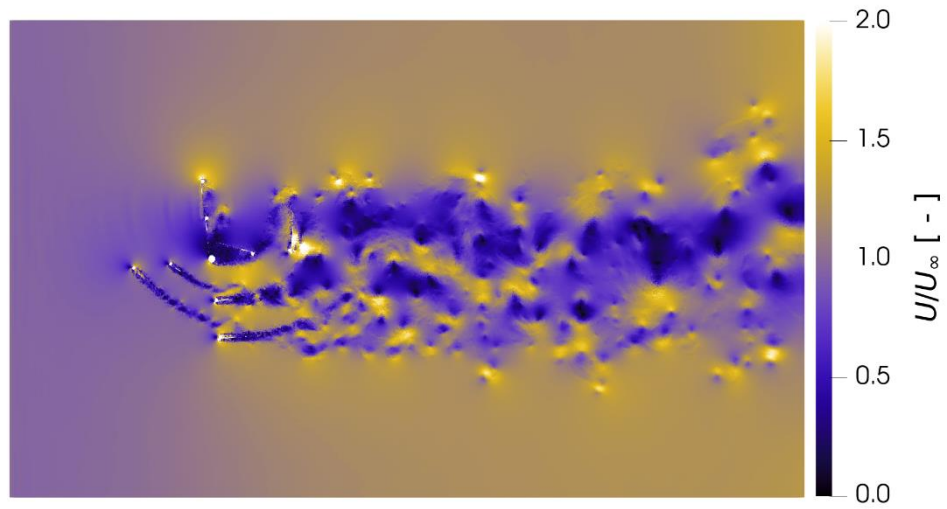


Fig. 15 Base condition flow field ($U_\infty = 0.9$ m/s) at $t = T$

439

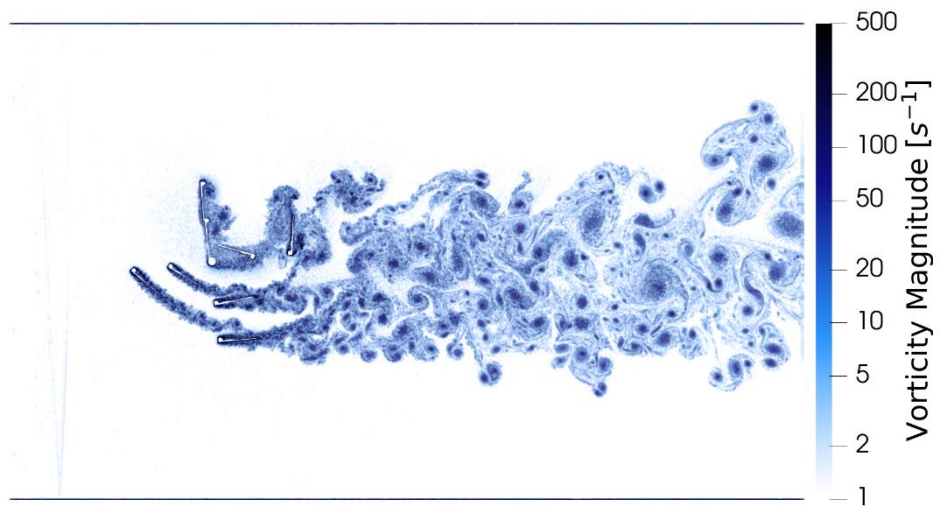


Fig. 16 Base condition Vorticity field at $t = T$, at $U_\infty = 0.9$ m/s

440

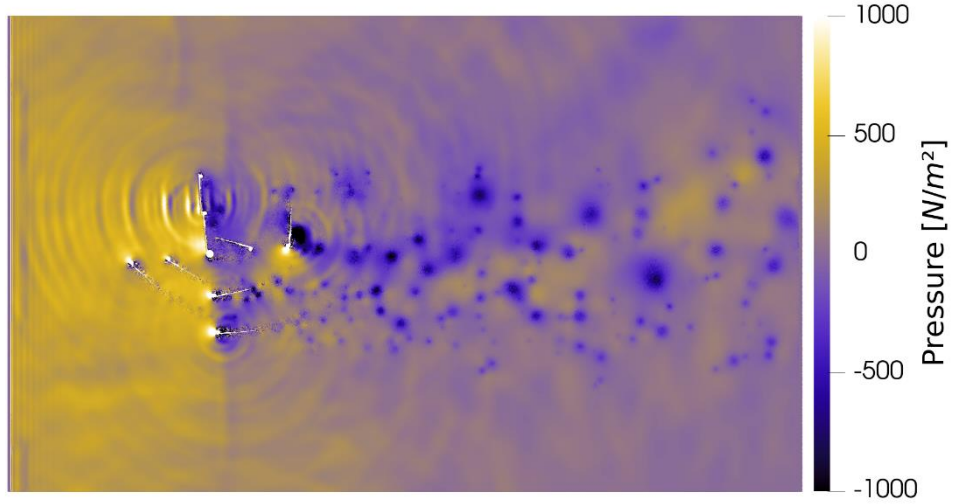


Fig. 17 Base condition pressure field at $t = T$, at $U_\infty = 0.9$ m/s

3.2 VALIDATION

As turbines of this design are certain to be operating in a variety of conditions, it is important not only that they can operate in a variety of flows, but that their performance in these flows is predictable. Prior tests with the CarBine were carried out at U_∞ of 0.51, 0.64, 0.77 and 1.08 m/s in fully submerged, free-spin conditions. These conditions were assessed in the SPH model and additionally provide validation of the degree to which the SPH model is able to represent the device operation beyond the base condition, which was selected due to being identified as a point of optimal performance in flume testing [15].

The angular velocity distributions for this set of tests are shown in Table 1; along with the error associated to each inlet velocity. Here it can be seen that the SPH model had a tendency to overestimate device performance (as was found to generally be the case for 2D models by Ouro and Stoesser [20]), with the error growing as the validation conditions become less similar to the base condition. The agreement of the SPH model to the laboratory data is not solely dependent on the flow velocity of the test, but may be less reliable when the flow speeds are farther from the optimal condition, and thus the motion of the device itself is less reliable.

Table 1: Mean angular velocity at varying flow speeds

U_∞ [m/s]	0.51	0.64	0.77	1.08
SPH [rad/s]	2.191	2.741	3.138	4.540
Flume [rad/s]	1.991	2.656	3.182	4.321
L2 Error Norm [-]	0.100	0.032	0.014	0.051

This difference in velocities can be seen in the angular displacement of the turbine during these flow conditions, as shown in Fig. 18. This shows the displacement of the SPH model as compared with the measured rotation from the flume plotted against the absolute difference between the incident inflow velocity and the base inflow velocity. Thus, the ability of the model to represent the operation of the turbine at a variety of flow states was confirmed.

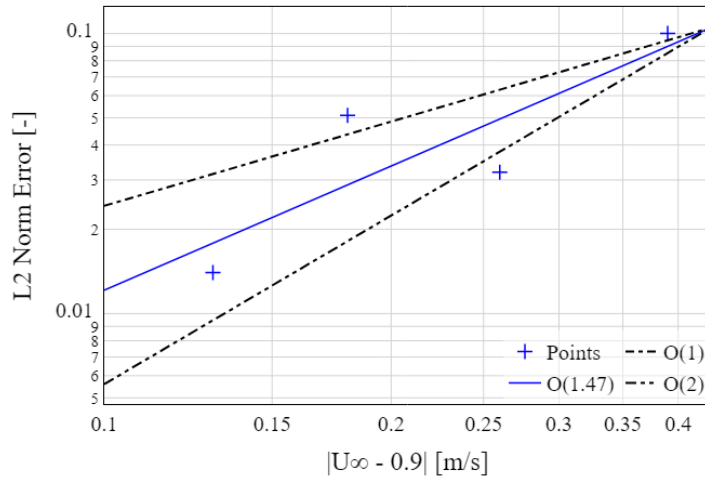


Fig. 18 Comparison of lab and SPH performance at multiple flow velocities

The tip speed ratio at the varied flow rates is shown in Fig. 19, with the base condition underlaid. It can be seen that the turbine behaviour was consistent with different flow ranges. There is an increased level of variability at the lowest flow velocity; this may be because the turbine was entering the lower operating range where the behaviour could be less predictable and more difficult to simulate.

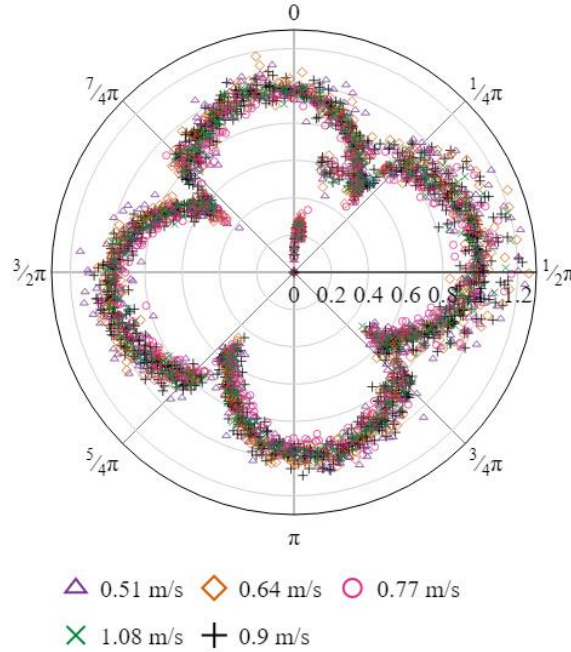


Fig. 19 Tip speed ratio at varied stream flow velocity

3.3 POST-DAMAGE RESILIENCE

The two damage cases are noticeably more varied in their angular velocity (shown in Fig. 20), with the periods when the missing flap is driving the device exhibiting a lower rotational velocity and so a lower net displacement. The performance of the models suggests that the loss of an outer arm (Case

2) is more impactful, and thus the outer arms would be better suited to reinforcement or more durable materials. This was to be expected on an empirical basis as the outer arm has a greater moment contribution to the device for the same force magnitude due to being farther from the axis. Further analysis could be carried out using this approach to assess the various combinations of lost flaps to determine ideal and worst-case layout combinations for a more severely damaged device.

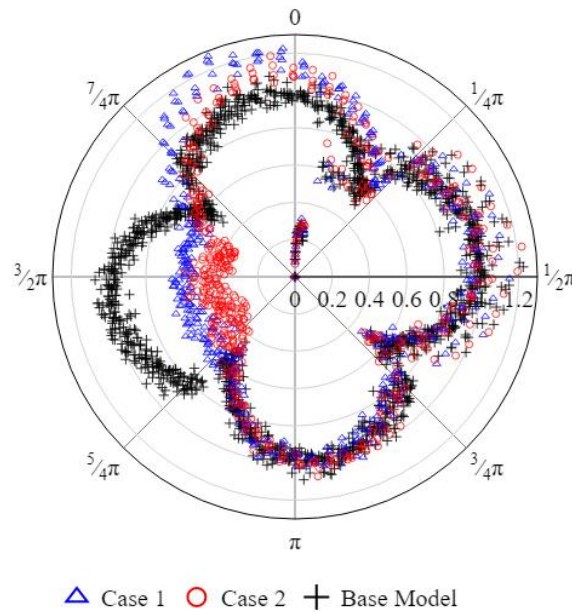


Fig. 20 Tip speed ratio comparison of damage options, at $U_\infty = 0.9$ m/s

4. CONCLUSIONS

To assess the potential of Smoothed Particle Hydrodynamics to the modelling of a vertical axis turbine and demonstrate this by providing some initial design insights, a vertical axis turbine design was represented in a two-dimensional simulation under a variety of flows and physical conditions. A series of experiments were conducted in the laboratory and used to support calibration and validation of the model. The simulations showed reasonable agreement with the flume results of the base design at multiple flow velocities. The primary assessment used the displacement of the device, as this was not prescribed in the SPH model, a key novelty of the method. The base design model was validated at a variety of flow states, demonstrating the ability of the SPH model to represent the free-spinning motion of a vertical axis turbine without the need for a prescribed rotation, thus ensuring the device is truly driven by the flow and is able to demonstrate hydrodynamically driven self-start. The model was highly prone to the formation of voids, thus a large shifting rate, and the artificial viscosity formulation were required to mitigate this. The model converged at a greater than second-order rate to the particle spacing, and kernel support, and was insensitive to the position of the downstream boundary. Two post-damage cases were tested to discern how the turbine would operate if it were required to produce energy once damaged; a motion state that was previously unknown (including hydrodynamically driven self start). All tests were run without power take-off and the effect of extracting energy from the device would certainly alter the motion and is an aspect of the device that could be investigated in future work. The errors associated with using a 2D simulation should be noted, and the process established here used to progress this method to 3D models when relevant effects are to be assessed. The SPH method was able to represent the vertical axis turbine and then provide design inputs to develop aspects of the design.

COMPLIANCE WITH ETHICAL STANDARDS:

Conflict of Interest: The authors declare that they have no conflict of interest.

Funding: This work was funded as part of the Water Informatics Science and Engineering Centre for Doctoral Training (WISE CDT) under a grant from the Engineering and Physical Sciences Research Council (EPSRC), grant number EP/L016214/1.

Ethical approval: This article does not contain any studies with human participants or animals performed by any of the authors.

Informed consent: Informed consent was obtained from all individual participants included in the study.

REFERENCES

1. UNFCCC. Secretariat (2021) Nationally determined contributions under the Paris Agreement - Synthesis report. Glasgow
2. The Crown Estate (2012) UK Wave and Tidal Key Resource Areas Project - Summary Report. *The Crown Estate* 1–10
3. Smart G, Noonan M (2018) Tidal Stream and Wave Energy Cost Reduction and Industrial Benefit: Summary Analysis. Report by ORE Catapult
4. Kirke B (2019) Hydrokinetic and ultra-low head turbines in rivers: A reality check. *Energy Sustain Dev* 52:1–10. <https://doi.org/10.1016/j.esd.2019.06.002>
5. Pudjianto D, Frost C, Coles D, Angeloudis A, Smart G, Strbac G (2023) UK Studies on the Wider Energy System Benefits of Tidal Stream. *Energy Adv* 2:12–33. <https://doi.org/10.1039/D2YA00251E>
6. Ridgill M, Neill SP, Lewis MJ, Robins PE, Patil SD (2021) Global riverine theoretical hydrokinetic resource assessment. *Renew Energy* 174:654–665. <https://doi.org/10.1016/j.renene.2021.04.109>
7. Kirke B (2020) Hydrokinetic turbines for moderate sized rivers. *Energy Sustain Dev* 58:182–195. <https://doi.org/10.1016/j.esd.2020.08.003>
8. ABP Marine Environmental Research Ltd (2008) Atlas of UK Marine Renewable Energy Resources: Technical Report
9. Office for National Statistics (2011) 2011 UK Census
10. Khan MJ, Bhuyan G, Iqbal MT, Quaicoe JE (2009) Hydrokinetic energy conversion systems and assessment of horizontal and vertical axis turbines for river and tidal applications: A technology status review. *Appl Energy* 86:1823–1835. <https://doi.org/10.1016/j.apenergy.2009.02.017>
11. Enevoldsen P, Xydis G (2019) Examining the trends of 35 years growth of key wind turbine components. *Energy Sustain Dev* 50:18–26. <https://doi.org/10.1016/j.esd.2019.02.003>
12. Kumar Y, Ringenberg J, Depuru SS, Devabhaktuni VK, Lee JW, Nikolaidis E, Andersen B, Afje A (2016) Wind energy: Trends and enabling technologies. *Renew Sustain Energy Rev* 53:209–224. <https://doi.org/10.1016/j.rser.2015.07.200>
13. Harries T, Kwan A, Brammer J, Falconer RA (2016) Physical testing of performance characteristics of a novel drag-driven vertical axis tidal stream turbine; With comparisons to a conventional Savonius. *Int J Mar Energy* 14:215–228. <https://doi.org/10.1016/j.ijome.2016.01.008>
14. Rauen W, Lin B, Falconer RA (2008) Transition from wavelets to ripples in a laboratory flume with a diverging channel. *Int J Sediment Res* 23:1–12. [https://doi.org/10.1016/S1001-6279\(08\)60001-3](https://doi.org/10.1016/S1001-6279(08)60001-3)
15. Ranabhat B (2022) Parametric Optimisation of a Vertical Axis Hydrokinetic Turbine with Shallow Water Application (PhD Thesis). Cardiff University
16. Maitre T, Amet E, Pellone C (2013) Modeling of the flow in a Darrieus water turbine: Wall grid refinement analysis and comparison with experiments. *Renew Energy* 51:497–512. <https://doi.org/10.1016/j.renene.2012.09.030>
17. Zidonis A, Benzon DS, Aggidis GA (2015) Development of hydro impulse turbines and new opportunities. *Renew Sustain Energy Rev* 51:1624–1635. <https://doi.org/10.1016/j.rser.2015.07.007>
18. Allmark MJ, Ellis R, Lloyd C, Ordonez-Sanchez S, Johannesen K, Byrne C, Johnstone C, O'Doherty T, Mason-Jones A (2020) The development, design and characterisation of a scale model Horizontal Axis Tidal Turbine for dynamic load quantification. *Renew Energy* 156:913–930. <https://doi.org/10.1016/j.renene.2020.04.060>
19. Lanzafame R, Mauro S, Messina M (2013) Wind turbine CFD modeling using a correlation-based

- transitional model. *Renew Energy* 52:31–39. <https://doi.org/10.1016/j.renene.2012.10.007>
20. Ouro P, Stoesser T (2017) An immersed boundary-based large-eddy simulation approach to predict the performance of vertical axis tidal turbines. *Comput Fluids* 152:74–87. <https://doi.org/10.1016/j.compfluid.2017.04.003>
21. Salehi S, Nilsson H, Lillberg E, Edh N (2021) Numerical Simulation of Hydraulic Turbine during Transient Operation Using OpenFOAM. *IOP Conf Ser Earth Environ Sci* 774:. <https://doi.org/10.1088/1755-1315/774/1/012060>
22. Yang B, Lawn C (2011) Fluid dynamic performance of a vertical axis turbine for tidal currents. *Renew Energy* 36:3355–3366. <https://doi.org/10.1016/j.renene.2011.05.014>
23. Hashemi SM, Moghimi M, Derakhshan S (2020) Experimental and numerical study of a flapping-blade vertical-axis hydrokinetic turbine under free surface deformation and blockage effects. *Int J Environ Sci Technol* 17:3633–3650. <https://doi.org/10.1007/s13762-020-02642-y>
24. Gingold RA, Monaghan J (1977) Smoothed particle hydrodynamics: theory and application to non-spherical stars. *Mon Not R Astron Soc* 181:541–559
25. Lucy L (1977) A numerical approach to the testing of the fission hypothesis. *Astron J* 82:1013–1024
26. Majtan E, Cunningham LS, Rogers BD (2021) Flood-Induced Hydrodynamic and Debris Impact Forces on Single-Span Masonry Arch Bridge. *J Hydraul Eng* 147:1–15. [https://doi.org/10.1061/\(asce\)hy.1943-7900.0001932](https://doi.org/10.1061/(asce)hy.1943-7900.0001932)
27. Altomare C, Crespo AJ., Domínguez JM, Gómez-Gesteira M, Suzuki T, Verwaest T (2015) Applicability of Smoothed Particle Hydrodynamics for estimation of sea wave impact on coastal structures. *Coast Eng* 96:1–12. <https://doi.org/10.1016/j.coastaleng.2014.11.001>
28. Baines A, Watson P, Cunningham LS, Rogers BD, Murphy J, Lizondo S (2019) Modelling shore-side pressure distributions from violent wave breaking at a seawall. *Proc Inst Civ Eng - Eng Comput Mech* 172:118–123. <https://doi.org/10.1680/jencm.19.00007>
29. Altomare C, Gironella X, Crespo AJ. (2021) Simulation of random wave overtopping by a WCSPH model. *Appl Ocean Res* 116:102888. <https://doi.org/10.1016/j.apor.2021.102888>
30. Altomare C, Tafuni A, Domínguez JM, Crespo AJ., Gironella X, Sospedra J (2020) SPH simulations of real sea waves impacting a large-scale structure. *J Mar Sci Eng* 8:1–21. <https://doi.org/10.3390/jmse8100826>
31. Fourtakas G, Rogers BD (2016) Modelling multi-phase liquid-sediment scour and resuspension induced by rapid flows using Smoothed Particle Hydrodynamics (SPH) accelerated with a Graphics Processing Unit (GPU). *Adv Water Resour* 92:186–199. <https://doi.org/10.1016/j.advwatres.2016.04.009>
32. Xenakis AM, Lind SJ, Stansby PK, Rogers BD (2017) Landslides and tsunamis predicted by incompressible smoothed particle hydrodynamics (SPH) with application to the 1958 Lituya Bay event and idealized experiment. *Proc R Soc A Math Phys Eng Sci* 473:. <https://doi.org/10.1098/rspa.2016.0674>
33. Zubeldia EH, Fourtakas G, Rogers BD, Farias MM (2018) Multi-phase SPH model for simulation of erosion and scouring by means of the shields and Drucker–Prager criteria. *Adv Water Resour* 117:98–114. <https://doi.org/10.1016/j.advwatres.2018.04.011>
34. Tagliafierro B, Mancini S, Ropero-Giralda P, Domínguez JM, Crespo AJ., Viccione G (2021) Performance assessment of a planing hull using the smoothed particle hydrodynamics method. *J Mar Sci Eng* 9:1–19. <https://doi.org/10.3390/jmse9030244>
35. Tagliafierro B, Karimirad M, Martínez-Estévez I, Domínguez JM, Viccione G, Crespo AJ. (2022) Numerical Assessment of a Tension-Leg Platform Wind Turbine in Intermediate Water Using the Smoothed Particle Hydrodynamics Method. *Energies* 15:. <https://doi.org/10.3390/en15113993>
36. Tagliafierro B, Martínez-Estévez I, Domínguez JM, Crespo AJ., Göteman M, Engström J, Gómez-Gesteira M (2022) A numerical study of a taut-moored point-absorber wave energy converter with a linear power take-off system under extreme wave conditions. *Appl Energy* 311:118629. <https://doi.org/10.1016/j.apenergy.2022.118629>
37. Shadloo MS, Oger G, Le Touzé D (2016) Smoothed particle hydrodynamics method for fluid flows, towards industrial applications: Motivations, Current state, And challenges. *Comput Fluids* 136:11–34. <https://doi.org/10.1016/j.compfluid.2016.05.029>
38. Marongiu J-C, Leboeuf F, Parkinson E (2007) Numerical simulation of the flow in a Pelton turbine using the meshless method smoothed particle hydrodynamics: A new simple solid boundary treatment. *Proc Inst Mech Eng Part A J Power Energy* 221:849–856.

- <https://doi.org/10.1243/09576509JPE465>
39. Marongiu J-C, Leboeuf F, Caro J, Parkinson E (2010) Free surface flows simulations in Pelton turbines using an hybrid SPH-ALE method. *J Hydraul Res* 48:22–1686. <https://doi.org/10.1080/00221686.2010.9641244>
 40. Rentschler M, Marongiu J-C, Neuhauser M, Parkinson E (2018) Overview of SPH-ALE applications for hydraulic turbines in ANDRITZ hydro. *J Hydrodyn* 30:114–121. <https://doi.org/10.1007/s42241-018-0012-y>
 41. Mazhar H, Heyn T, Pazouki A, Melanz D, Seidl A, Bartholomew A, Tasora A, Negrut D (2013) CHRONO: A parallel multi-physics library for rigid-body, flexible-body, and fluid dynamics. *Mech Sci* 4:49–64. <https://doi.org/10.5194/ms-4-49-2013>
 42. Hansen JT, Mahak M, Tzanakis I (2021) Numerical modelling and optimization of vertical axis wind turbine pairs: A scale up approach. *Renew Energy* 171:1371–1381. <https://doi.org/10.1016/j.renene.2021.03.001>
 43. Roy S, Saha UK (2013) Numerical investigation to assess an optimal blade profile for the drag based vertical axis wind turbine. In: ASME International Mechanical Engineering Congress and Exposition, Proceedings (IMECE). pp 1–9
 44. Domínguez JM, Fourtakas G, Altomare C, Canelas R, Tafuni A, García-Feal O, Martínez-Estévez I, Mokos A, Vacondio R, Crespo AJ., Rogers BD, Stansby PK, Gómez-Gesteira M (2021) DualSPHysics: from fluid dynamics to multiphysics problems. *Comput Part Mech* 9:867–895. <https://doi.org/10.1007/s40571-021-00404-2>
 45. Harada T, Koshizuka S, Kawaguchi Y (2007) Smoothed particle hydrodynamics on GPU computing. *Proc Comput Graph Int* 40:63–70
 46. Martínez-Estévez I, Domínguez JM, Tagliaferro B, Canelas R, García-Feal O, Crespo AJ., Gómez-Gesteira M (2023) Coupling of an SPH-based solver with a multiphysics library. *Comput Phys Commun* 283:108581. <https://doi.org/10.1016/j.cpc.2022.108581>
 47. Monaghan J (2005) Smoothed particle hydrodynamics. *Reports Prog Phys* 68:1703–1759. <https://doi.org/10.1088/0034-4885/68/8/R01>
 48. Fourtakas G, Domínguez JM, Vacondio R, Rogers BD (2019) Local uniform stencil (LUST) boundary condition for arbitrary 3-D boundaries in parallel smoothed particle hydrodynamics (SPH) models. *Comput Fluids* 190:346–361. <https://doi.org/10.1016/j.compfluid.2019.06.009>
 49. Lind SJ, Xu R, Stansby PK, Rogers BD (2012) Incompressible smoothed particle hydrodynamics for free-surface flows: A generalised diffusion-based algorithm for stability and validations for impulsive flows and propagating waves. *J Comput Phys* 231:1499–1523. <https://doi.org/10.1016/j.jcp.2011.10.027>
 50. Skillen A, Lind S, Stansby PK, Rogers BD (2013) Incompressible smoothed particle hydrodynamics (SPH) with reduced temporal noise and generalised Fickian smoothing applied to body-water slam and efficient wave-body interaction. *Comput Methods Appl Mech Eng* 265:163–173. <https://doi.org/10.1016/j.cma.2013.05.017>
 51. Tafuni A, Domínguez JM, Vacondio R, Crespo AJ. (2018) A versatile algorithm for the treatment of open boundary conditions in Smoothed particle hydrodynamics GPU models. *Comput Methods Appl Mech Eng* 342:604–624. <https://doi.org/10.1016/j.cma.2018.08.004>
 52. English A, Domínguez JM, Vacondio R, Crespo AJ., Stansby PK, Lind SJ, Chiapponi L, Gómez-Gesteira M (2021) Modified dynamic boundary conditions (mDBC) for general-purpose smoothed particle hydrodynamics (SPH): application to tank sloshing, dam break and fish pass problems. *Comput Part Mech* 9:1–15. <https://doi.org/10.1007/s40571-021-00403-3>
 53. Crespo AJC, Gómez-Gesteira M, Dalrymple RA (2007) Boundary conditions generated by dynamic particles in SPH methods. *Comput Mater Contin* 5:173–184. <https://doi.org/10.3970/cmc.2007.005.173>
 54. Leimkuhler B, Matthews C (2015) Molecular Dynamics With Deterministic and Stochastic Numerical Methods. Springer
 55. Wendland H (1995) Piecewise polynomial, positive definite and compactly supported radial functions of minimal degree. *Adv Comput Math* 4:389–396. <https://doi.org/10.1007/BF02123482>
 56. Molteni D, Colagrossi A (2009) A simple procedure to improve the pressure evaluation in hydrodynamic context using the SPH. *Comput Phys Commun* 180:861–872. <https://doi.org/10.1016/j.cpc.2008.12.004>
 57. Sun PN, Colagrossi A, Marrone S, Antuono M, Zhang AM (2018) Multi-resolution Delta-plus-SPH

- with tensile instability control: Towards high Reynolds number flows. *Comput Phys Commun* 224:63–80. <https://doi.org/10.1016/j.cpc.2017.11.016>
58. Gotoh H, Shibahara T, Sakai T (2001) Sub-particle-scale turbulence model for the MPS method-Lagrangian flow model for hydraulic engineering. *Comput Fluid Dyn J* 9:339–347
 59. Gotoh H, Shao S, Memita T (2003) SPH-LES MODEL FOR NUMERICAL INVESTIGATION OF WAVE INTERACTION WITH PARTIALLY IMMERSED BREAKWATER. *Coast Eng J* 46:39–63
 60. Dalrymple RA, Rogers BD (2006) Numerical modeling of water waves with the SPH method. *Coast Eng* 53:141–147. <https://doi.org/10.1016/j.coastaleng.2005.10.004>
 61. Gómez-Gesteira M, Rogers BD, Dalrymple RA, Crespo AJ. (2010) State-of-the-art of classical SPH for free-surface flows. *J Hydraul Res* 48:6–27. <https://doi.org/10.1080/00221686.2010.9641242>
 62. Valizadeh A, Monaghan J (2012) Smoothed particle hydrodynamics simulations of turbulence in fixed and rotating boxes in two dimensions with no-slip boundaries. *Phys Fluids* 24:. <https://doi.org/10.1063/1.3693136>
 63. Robinson M, Monaghan J (2012) Direct numerical simulation of decaying two-dimensional turbulence in a no-slip square using smoothed particle hydrodynamics. *Int J Numer Methods Fluids* 70:37–55. <https://doi.org/10.1002/fld>
 64. Robinson M, Cleary P, Monaghan J (2008) Analysis of mixing in a Twin Cam Mixer using smoothed particle hydrodynamics. *AIChE J* 54:1987–1998. <https://doi.org/10.1002/aic.11530>
 65. Valizadeh A, Monaghan J (2015) SPH simulation of 2D turbulence driven by a cylindrical stirrer. *Eur J Mech - B/Fluids* 51:44–53. <https://doi.org/10.1016/j.euromechflu.2014.11.002>
 66. Morris JP, Monaghan J (1997) A switch to reduce SPH viscosity. *J Comput Phys* 136:41–50. <https://doi.org/10.1006/jcph.1997.5690>
 67. Quinlan NJ, Basa M, Lastiwka M (2006) Truncation error in mesh-free particle methods. *Int J Numer Methods Eng* 66:2064–2085. <https://doi.org/10.1002/nme.1617>