Impact of lateral turbine spacing on the performance of a multi-rotor tidal energy device

R. Smith, B. Townley, B. Wray, A. Mason-Jones

Abstract— **The impact of local blockage on the power production of a tidal array due to multiple turbines positioned in close proximity is studied. A numerical model of the HydroWing tidal energy device, which features multiple turbines on a retrievable wing, is being developed using geometry-resolved computational fluid dynamics (CFD). For this paper, the influence of two turbines at several spacings is considered. The CFD modelling is validated against a blade element momentum theory (BEMT) model and measured full scale data, and is then used to perform quasi-static steady-state simulations of two turbines in a twin rotor configuration, where a multiple reference frame (MRF) approach is used to simulate rotor rotation. Transient simulations are also conducted using a sliding mesh approach to study cyclic interaction between closely spaced turbines as they rotate. The lateral spacing between the rotors is varied and the resulting impact on the axial loads and power performance of the two turbines is studied, with the aim of identifying optimal turbine spacing for the HydroWing device. It is found that the optimal spacing between turbines for maximum power is also dependent on proximity to the seabed. The results will be used in future design optimisation work to minimize the levelized cost of energy of a large scale array using HydroWing technology at the proposed site for the Morlais tidal energy project.**

Keywords— **Tidal turbine, Tidal array, CFD, Blockage, Spacing**

This work was supported by the Marine Energy Engineering Centre of Excellence (MEECE) delivered by Offshore Renewable Energy Catapult (email: meece@ore.catapult.org.uk), and funded by the Welsh European Funding Office (WEFO)

R. Smith is with Cardiff University School of Engineering, Queen's Buildings, Cardiff CF24 3AA, UK (e-mail: smithr70@cardiff.ac.uk).

B. Townley is an IDCORE Research Engineer sponsored by HydroWing Ltd, Unit 3 Penstraze Business Centre, Penstraze, Truro, UK, TR4 8PN (e-mail: bryn.townley@ed.ac.uk).

B. Wray is with HydroWing Ltd (email: brw@inyanga.tech)

A. Mason-Jones is with Cardiff University School of Engineering (email: mason-jonesa@cardiff.ac.uk).

Digital Object Identifier: https://doi.org/10.36688/ewtec-2023-334

I. Introduction

PREVIOUS theoretical and experimental work has established that tidal turbines positioned close to the seabed or sea surface, or turbines closely packed in arrays can experience improved efficiency due to high local blockage causing flow acceleration between turbines and channel boundaries. The effects of partial blockage can result in tidal turbines exceeding the theoretical limit of efficiency (Lanchester-Betz limit) in unbounded flow [1], [2]. Nishino and Willden [3] demonstrated theoretically that the efficiency of a tidal array in an infinitely wide channel is maximised at an optimal local blockage ratio, which is a function of the rotor diameter, channel depth and spacing between turbines. However, turbines packed too closely together will 'choke' the flow and decrease turbine efficiency due to less flow through the array. An experimental study by Myers and Bahaj [4] demonstrated that for a sufficiently small lateral spacing, the flow between turbines is accelerated, and that a turbine placed downstream may produce higher power than the upstream turbines. Field tests conducted by Jeffcoate et al. [5] revealed that the power coefficient C_p of two turbines is dependent on their position relative to each other and their tip speed ratio (TSR), with a lateral spacing of two rotor diameters between turbine centrelines corresponding to an increase of up to 6% compared to the individual turbine.

HydroWing has proposed a novel tidal energy device concept that aims to address some of the key challenges currently facing tidal energy development, including low reliability and high costs associated with operations and maintenance (O&M). The design features rows of multiple rotors mounted on retrievable wings, which are installed on a modular frame foundation as illustrated in Fig. 1. The wings are vertically staggered to form flow corridors that prevent wake interaction. The design also features vertical lift corridors to enable easy installation and recovery of wings. The concept offers advantages in terms of reliability, reduced weight and simplified O&M. HydroWing is also a berth holder at the Morlais tidal stream energy project site off the coast of Holy Island, Anglesey, where an array with up to 30 MW capacity will be installed [6]. In order to develop a cost effective array with reliable energy production, the positioning of multiple turbines must be carefully considered.

In this work, the impact of blockage due to turbine spacing and seabed proximity on the performance of the turbine used in the HydroWing multi-rotor device is studied using a computational fluid dynamics (CFD) approach. The model is used to study the flow field and hydrodynamic loads on two turbines positioned in a row where the lateral spacing between them is varied. The aim of the study is to determine the dependence of the turbine performance on the local blockage conditions, and to identify the optimal spacing between rotors for maximum device efficiency. The results of this work will improve understanding of the performance of the HydroWing device, and will inform the design of an optimised array at the Morlais tidal project site to ensure efficient turbine operation and minimal levelized cost of energy (LCOE).

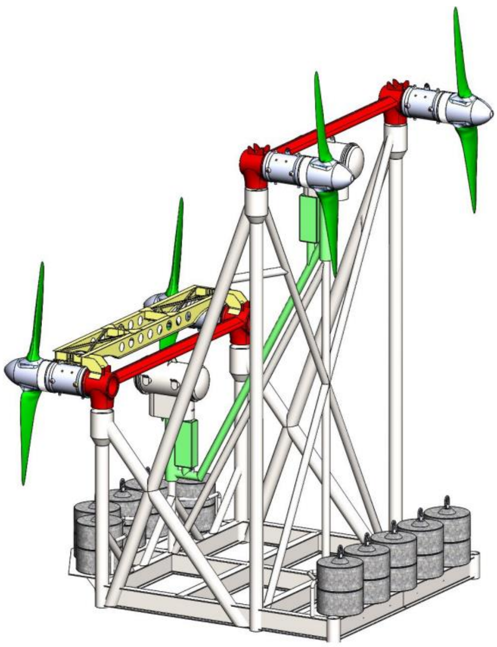


Fig. 1. HydroWing multi-rotor device concept

II. CFD MODELLING AND VALIDATION

The turbine studied in this work is a horizontal axis, two bladed, bidirectional design from the T-2si series developed by Tocardo, and has a rated power of 250 kW. Geometry-resolved CFD models of the turbine rotor are developed in ANSYS Fluent and Star-CCM+.

Steady state calculations of the flow field are performed by simulating rotor rotation using a multiple reference frame (MRF) approach. Each turbine is located inside a cylindrical rotating mesh zone with a 1.05D diameter, where D is the turbine diameter. To isolate turbine performance, the support structure was not included at this stage of the study.

Computational meshes consisting of polyhedral cells are generated for both ANSYS Fluent and Star-CCM+ models. In the steady state calculations, the mesh generated for each MRF region containing a turbine consists of approximately 5.2 million cells. The mesh features increased refinement in the blade tip regions and 10 layers of boundary cells at the turbine wall in order to capture boundary layer effects. The cell thickness at the wall is 0.2 mm. The mesh is shown in Fig. 2.

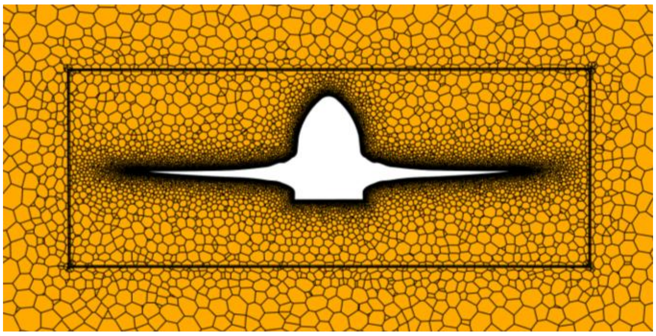
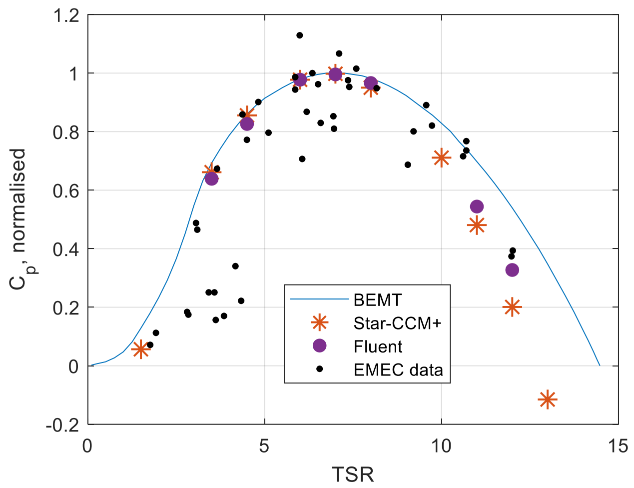


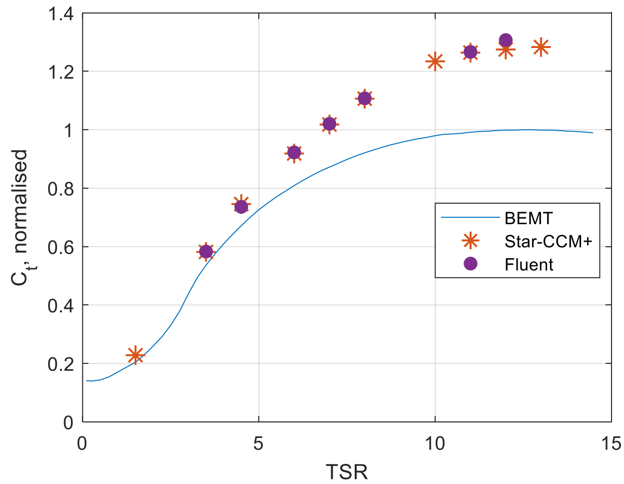
Fig. 2. Computational mesh of the Tocardo turbine in ANSYS Fluent

The CFD modelling in this work uses a Reynold's Averaged Navier Stokes (RANS) approach with the k-ω SST turbulence model. A uniform inflow velocity of 2 m/s is specified across the inlet boundary in all cases. The turbines are situated 5D downstream from the inlet, and the turbulence intensity at the rotor location is approximately 3%.

The CFD modelling is validated against an in-house blade element momentum theory (BEMT) model from Tocardo and measured data for a full scale Tocardo T2 turbine deployed at the European Marine Energy Centre (EMEC). The validation process involved performing simulations of one turbine operating in freestream flow at a range of different constant rotational speeds. The resulting hydrodynamic performance data from the numerical modelling and experimental data is shown in Fig. 3. The power resulting from the two CFD models is shown to agree well with each other as well as with the BEMT model, particularly close to the peak of the power curve. The measured data has a high degree of scatter, as expected from an onsite turbine, however the trend is of similar magnitude and trend as the simulated curves. The CFD models predict earlier stalling than the BEMT model, which results in divergence between the models at higher tip speed ratios.



(a) Normalized power coefficient, C_p



(b) Normalized thrust coefficient, C_t

Fig. 3. (a) Power and (b) thrust data at different tip speed ratio values for the Tocardo turbine

III. UNIFORM FLOW

Simulations in freestream flow conditions are conducted, where the domain boundaries are placed far away from the turbines so that they can be considered to have negligible influence on performance. The two turbines are placed in a rectangular domain where each of the walls are assigned free-slip conditions. The inflow velocity is constant with both depth and width across the domain. The lateral spacing between the blade tips of the two turbines is varied from 0.15D to 4D. For the rest of this paper, the presented results are obtained using the model developed in ANSYS Fluent only; the results of the Star-CCM+ model are presented in a separate publication.

A. Rotor loads

The rotor loads for the two turbines in the free stream are shown in Fig. 4. In these simulations, the two turbines rotate in the same direction (counter-clockwise when viewed from upstream), at a constant rotational speed to give a TSR of 7, which represents the peak of the power curve for a single turbine operating in freestream flow as shown in Fig. 3.

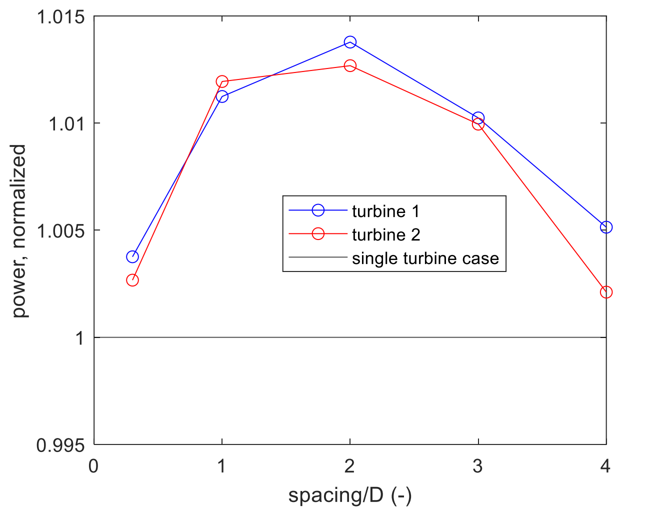
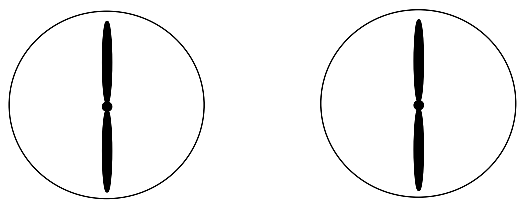


Fig. 4. Rotor power variation with lateral spacing for freestream flow conditions.

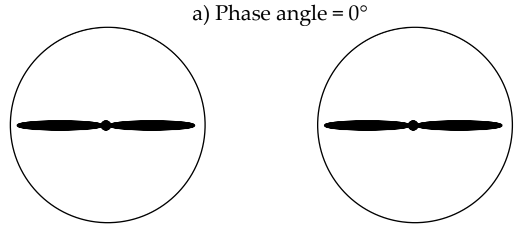
The quasi-steady state MRF simulations of two turbines show that the lateral spacing has a small impact on the power production. All cases involving two turbines appear to show a small increase in power compared to the single turbine calculation, with a maximum increase of less than 1.5%.

TABLE I
POWER OUTPUT OF TWO TURBINES IN DIFFERENT STEADY STATE SIMULATION CONFIGURATIONS, TIP SPACING = 1D. VALUES LISTED ARE AVERAGE OF BOTH TURBINES

	Change in power per turbine relative to single turbine case (%)
Vertical blades	+ 1.16
Horizontal blades	+ 0.96
Horizontal blades, counter rotation	+ 1.21



a) Phase angle = 0°



b) Phase angle = 90°

Fig. 5. Blades at a) 0° and b) 90° phase

The rotor loads for different case studies involving a rotor spacing of 1D are listed in Table I. These cases include blades at different phase angles. When an MRF approach is used, the turbines do not physically rotate within the fluid, and the quasi-steady simulations instead provide a 'snapshot' of the fluid flow at specified moments in time. Therefore, in order to investigate how the interaction between turbines may vary as they rotate, quasi-steady calculations of the turbines at relative blade phase angles of 0° and 90° are performed, where the blades are pointed vertically and horizontally respectively as illustrated in Fig. 5. The turbine loads are found to be marginally higher when the blades are vertical, it should be noted that there is a uniform flow upstream of the rotor. The impact of counter-rotating rotors is also studied. Counter-rotating turbines have a potential advantage in terms of the structural response of the multi-rotor device, since they will have opposing torque that will cancel each other out, therefore potentially reducing the cyclic loading and subsequent fatigue on the support structure. However, they have a disadvantage in practice, since they will require different blade molds, therefore increasing initial costs. The results presented in Table I reveal that counter rotating turbines make little difference to the power output compared to same-direction rotating turbines.

B. *Flow acceleration and turbine wake*

Fig. 6 shows the influence of the turbine spacing on the velocity of the flow passing between them, where U/U_0 is the axial velocity normalized by the freestream velocity value. The blockage effect causes the fluid flow to accelerate immediately downstream from the turbines, and this acceleration increases as the spacing is reduced. However, for small spacings, the velocity drops significantly at a short distance downstream from the turbines as the two wakes combine. This is illustrated further in Fig. 7.

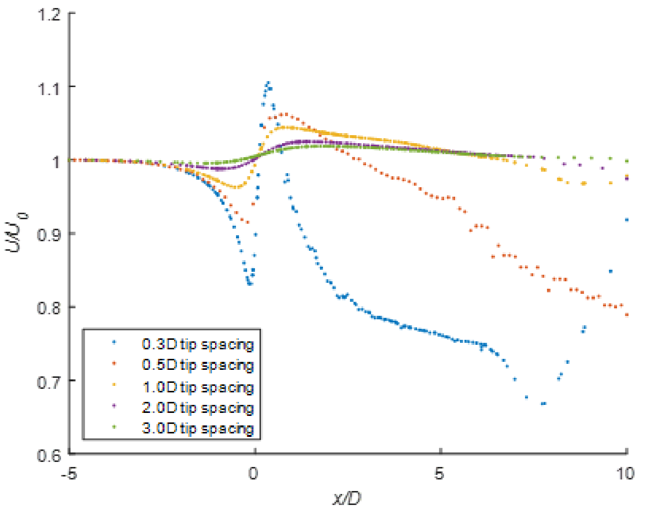


Fig. 6. Axial flow velocity between two turbines with varying lateral spacing. The turbines are positioned at x = 0

The flow acceleration between turbines indicates that a turbine placed a certain distance downstream and laterally offset from the two upstream turbines may experience a small improvement in efficiency due to the higher incoming flow velocity. This would likely require a spacing of 2D or greater between the two upstream turbines in order to ensure the downstream turbine is not operating in wake flow. For a simulated tip spacing of 2D, the maximum flow velocity due to acceleration between turbines is approximately 3% higher than the freestream flow. The optimal position for a downstream turbine depends on the spacing between the upstream turbines; the maximum flow velocity occurs further downstream as the spacing is increased. To illustrate this further, the wake velocity at different locations downstream from the two turbines with 2D spacing is plotted in Fig. 8, where x is the downstream distance from the turbines, each line in Fig. 8 is related to turbine lateral spacing, see legend. The two wakes are shown to widen as they propagate downstream. It should be noted that the accuracy of the wake modelling may be limited since the k-ω SST turbulence model used in this work assumes isotropic turbulence in the wake due to the use of the k-ϵ model in regions without proximity to a wall. Previous work [7]–[10] has demonstrated that the rotation of turbine blades induces significant anisotropy in the turbine wake, and isotropic turbulence models therefore result in underpredictions in the wake turbulence. Higher fidelity turbulence modelling such as LES may therefore be required to obtain a more reliable estimate of the wake behaviour and determine an optimal placement for a downstream turbine.

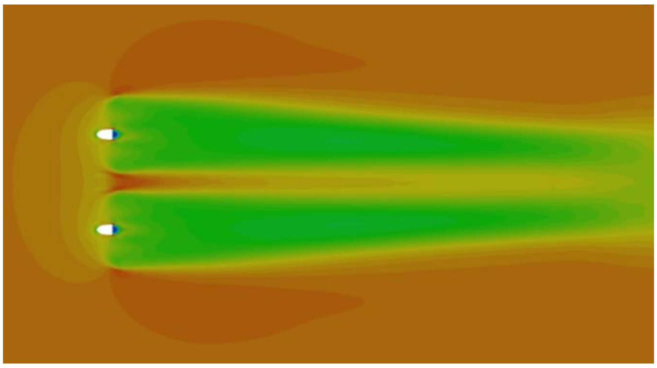


Fig. 7. Downstream merging of wakes of closely spaced turbines (tip spacing = 0.3D)

IV. TRANSIENT EFFECTS

A sliding mesh approach was used to investigate the transient behaviour of the two turbines interacting as they rotate. A coarser mesh was used for these simulations in order to reduce the computational cost. The mesh size for the rotating region containing the turbine is reduced to 1.7 million cells. The boundary layer cell thickness is also increased to 1 mm, which results in the maximum y+ value increasing from approximately 100 to over 300 for the simulated cases with a TSR of 7.

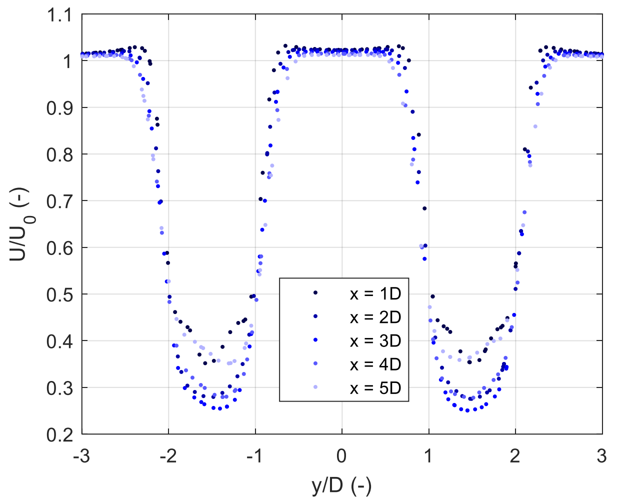
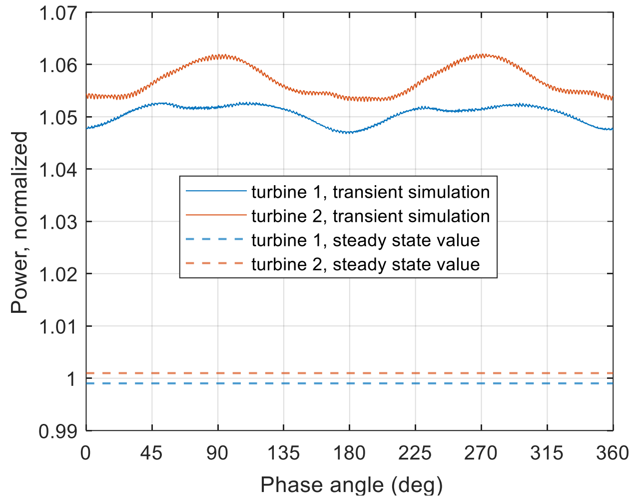


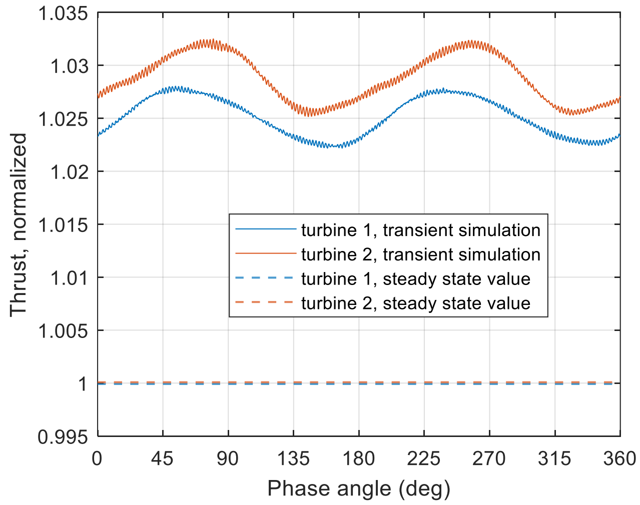
Fig. 8. Flow velocity downstream from two turbines with 2D tip spacing

The transient simulations are initialised by first running a steady-state calculation with the MRF approach. The reduction in mesh refinement results in an approximately 6% drop in the steady state rotor load values. The computational cost is considerably higher than that of the steady state simulations despite the reduced mesh size; the sliding mesh simulations were found to require a minimum of 12 turbine rotations before the rotor loads began to stabilize, which required a runtime of over 1 week running in parallel on 20 CPUs.

Cases with close spacing (0.3D) between turbines were selected for transient simulation in order to study possible interaction between rotors as they rotate. The variation in the rotor power and thrust force over the course of one rotor rotation are shown in Fig. 9 and Fig. 10 for same direction- and counter-rotating turbines respectively. There is cyclic variation, and a small difference between the loading for the two turbines. The peak loading occurs at close to 90° and 270° phase in both cases, which is when the blades are horizontal and the blade tips of the two turbines pass each other, however the cyclic variation is slightly different for each turbine. This is not consistent with the results of the steady state simulations presented in Table I, which indicated that turbine power is slightly higher when the blades are vertical. This suggests that the MRF approach may not fully capture the flow field around a rotating turbine. It can be observed that the power and thrust are out of phase with each other, where the thrust peaks earlier in the rotation cycle than the power. The peak to peak variation is small (less than 1% of the average value) in both cases. The results suggest that the relative rotational direction of the turbines (i.e. counter rotation vs same direction rotation) makes little difference to the loading on the rotors. It is noted that the average loads predicted by the transient simulations are higher than the calculated steady state values; the power is approximately 5% higher in both cases, and the thrust is around 2.5% higher. This again indicates possible limitations of the MRF approach.



(a) Normalized power



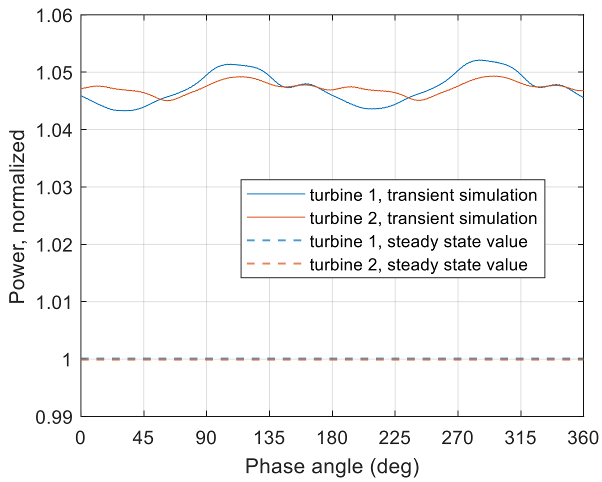
(b) Normalized thrust

Fig. 9. Variation in rotor (a) power and (b) thrust over one rotation cycle for two turbines rotating in the same direction, normalized by the steady state value (averaged between both turbines). Tip spacing between rotors = 0.3D, TSR = 7

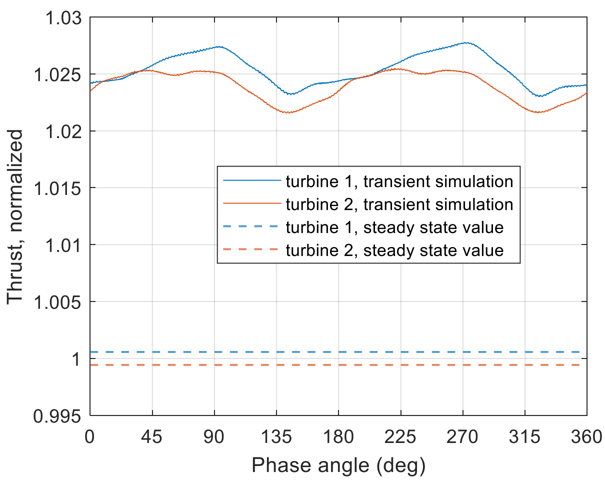
V. INFLUENCE OF THE SEABED

The impact of turbine proximity to the seabed on rotor performance is investigated. The seabed is modelled as a smooth wall with a no-slip boundary condition. The turbine hub height above the seabed is 12 m, resulting in a blade tip clearance distance of approximately 7 m. A computational mesh featuring 10 inflation layers at the seabed surface is used to ensure that the boundary layer is captured. The influence of water depth is not considered in this study in order to reduce simulation complexity and computational costs associated with modelling the free surface. The top and side boundaries are therefore once again modelled using free-slip boundary conditions. The inflow velocity profile is uniform except for close to the seabed boundary, where the no-slip condition takes effect. It should be noted that a more realistic representation of seabed influence on tidal flow would incorporate a velocity shear profile with increasing flow velocity with height. To maintain simplicity, this was not considered in this work as this would introduce additional cyclic load fluctuation with rotor rotation that would require transient simulation to capture. The impact of more realistic velocity profiles including velocity shear on turbine performance will be studied in future work.

A case involving one turbine is first simulated. The additional blockage due to the presence of the seabed causes flow acceleration between the rotor and seabed surface, and leads to a small increase in the hydrodynamic loads compared to the freestream case [2]. At a TSR of 7, for which maximum turbine efficiency is reached in free stream flow, the power coefficient increases by 3.4% compared to free stream flow.



(a) Normalized power



(b) Normalized thrust

Fig. 10. Variation in rotor (a) power and (b) thrust over one rotation cycle for two counter-rotating turbines, normalized by the steady state value (averaged between both turbines). Tip spacing between rotors = 0.3D, TSR = 7

A design optimisation study of laterally offset tidal turbines by Schluntz and Willden [11] found that the required tip speed ratio (TSR) for maximised C_p may increase as the blockage is increased. Therefore, the influence of TSR on Tocardo twin rotor performance is examined by also performing simulations at a higher TSR of 8. For the single turbine, a larger increase in C_p of 4% is observed at TSR = 8, however the power output is still lower than for TSR = 7.

Fig. 11 shows the variation in rotor power with spacing when the seabed is modelled. The turbines are rotating in the same direction in all cases. The largest power increase occurs at a tip spacing of around 1D between turbines; this increase is approximately 2% for a TSR of 7. The relative increase for TSR = 8 is again larger (approximately 4% for a 1D tip spacing), however the highest power is still obtained at TSR = 7. It can therefore be concluded that at the proposed turbine height from the seabed, the optimal tip speed ratio for the rotor does not change from the freestream value. The optimal turbine spacing for maximum power is lower when the seabed is modelled than in freestream flow, where the optimal spacing was around 2D as shown in Fig. 4. This highlights the importance of considering turbine proximity to the seabed when designing tidal arrays.

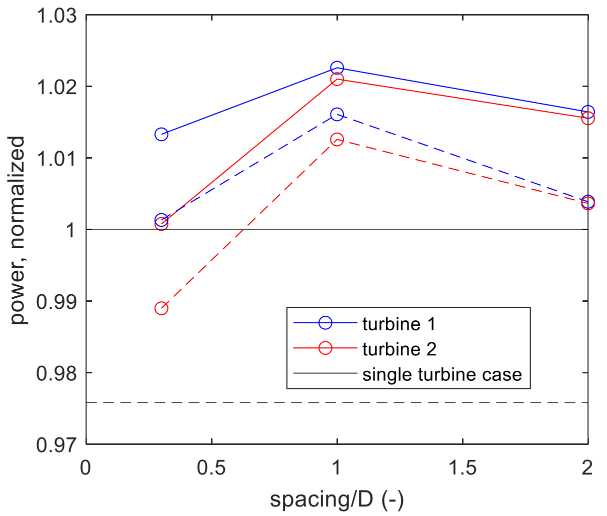


Fig. 11. Variation in rotor power with lateral spacing for turbines positioned close to the seabed, normalized by the power for a single turbine with TSR=7. Solid lines represent cases where TSR=7, dashed lines represent TSR=8

VI. CONCLUSION

In this work, the impacts of blockage on the Tocardo turbines used in the HydroWing device has been studied numerically using high fidelity CFD modelling, which has been validated against BEMT modelling and measured full scale data. With the caveat of realistic flows not being considered in this study, is noted that tidal turbines in arrays can produce more power than individual turbines due to increased local blockage resulting in acceleration of fluid flow. The presented analysis shows that the performance of a row of turbines is affected by the lateral spacing between rotors, where a power increase of approximately 2% per turbine can be obtained at an optimal spacing, which appears to be around 1 rotor diameter between the tips of the two rotors. It is also found that proximity to the seabed results in an increase in power performance compared to operation in freestream flow, with an observed 3.4% increase in the maximum C_p for a single turbine at a hub height of 12 m. Overall it is found that the effects of blockage on the Tocardo turbine and proposed HydroWing design are small, at least for a two-rotor case, however although the increase in power may be small as a percentage this increase can significantly influence the total annual energy extracted by each rotor. The impact of counter- versus same direction rotation is also examined, and found to have negligible impact on rotor loads.

Future work will study the performance of the HydroWing device in flow conditions specific to the Morlais tidal project site, using measured ADCP data. The outcomes of the presented analysis and future work will be used to design an optimised array for the proposed site using HydroWing technology.

ACKNOWLEDGEMENT

The presented research made use of high performance computing facilities provided by Advanced Research Computing at Cardiff (ARCCA).

REFERENCES

[1] R. Vennell, "Exceeding the Betz limit with tidal turbines," *Renewable Energy*, vol. 55, pp. 277–285, Jul. 2013, doi: 10.1016/j.renene.2012.12.016.

[2] C. Garrett and P. Cummins, "The efficiency of a turbine in a tidal channel," *Journal of Fluid Mechanics*, vol. 588, pp. 243–251, Oct. 2007, doi: 10.1017/S0022112007007781.

[3] T. Nishino and R. H. J. Willden, "The efficiency of an array of tidal turbines partially blocking a wide channel," *Journal of Fluid Mechanics*, vol. 708, pp. 596–606, Oct. 2012, doi: 10.1017/jfm.2012.349.

[4] L. E. Myers and A. S. Bahaj, "An experimental investigation simulating flow effects in first generation marine current energy converter arrays," *Renewable Energy*, vol. 37, no. 1, pp. 28–36, Jan. 2012, doi: 10.1016/j.renene.2011.03.043.

[5] P. Jeffcoate, T. Whittaker, C. Boake, and B. Elsaesser, "Field tests of multiple 1/10 scale tidal turbines in steady flows," *Renewable Energy*, vol. 87, pp. 240–252, Mar. 2016, doi: 10.1016/j.renene.2015.10.004.

[6] A. van Unen, "HydroWing to demonstrate next generation Tocardo Turbines at Morlais," *Tocardo*, Feb. 08, 2022. https://www.tocardo.com/hydrowing-to-demonstrate-next-generation-tocardo-turbines-at-morlais/ (accessed May 15, 2023).

[7] R. Ellis, "The influence of inter-device spacing on the performance of a tidal stream turbine," PhD, Cardiff University, 2020. Accessed: May 24, 2023. [Online]. Available: https://orca.cardiff.ac.uk/id/eprint/136449/

[8] T. Ebdon *et al.*, "The impact of turbulence and turbine operating condition on the wakes of tidal turbines," *Renewable Energy*, vol. 165, pp. 96–116, Mar. 2021, doi: 10.1016/j.renene.2020.11.065.

[9] M. Allmark *et al.*, "A detailed study of tidal turbine power production and dynamic loading under grid generated turbulence and turbine wake operation," *Renewable Energy*, vol. 169, pp. 1422–1439, May 2021, doi: 10.1016/j.renene.2020.12.052.

[10] S. C. Tedds, I. Owen, and R. J. Poole, "Near-wake characteristics of a model horizontal axis tidal stream turbine," *Renewable Energy*, vol. 63, pp. 222–235, Mar. 2014, doi: 10.1016/j.renene.2013.09.011.

[11] J. Schluntz and R. H. J. Willden, "The effect of blockage on tidal turbine rotor design and performance," *Renewable Energy*, vol. 81, pp. 432–441, Sep. 2015, doi: 10.1016/j.renene.2015.02.050.