Proceedings of the Thirty-fourth (2024) International Ocean and Polar Engineering Conference
Rhodes, Greece, June 16–21, 2024
Copyright © 2024 by the International Society of Offshore and Polar Engineers (ISOPE)
ISBN 978-1-880653-78-4; ISSN 1098-6189

www.isope.org

Numerical Wave Tank Development for Hydrodynamic Analysis of a Tidal Turbine

Kai Xu[1, 2], *William Finnegan*[1, 2], *Fergal O'Rourke*[3], *Jamie Goggins*[1, 2]
[1]School of Engineering, University of Galway, Galway, Ireland
[2] Construct Innovate and the SFI MaREI Research Centre, Ryan Institute, University of Galway, Galway, Ireland
[3]School of Engineering, Dundalk Institute of Technology, Dundalk, Ireland

ABSTRACT

Many existing studies show that the ocean tidal turbine experiences frequent and large-scale fluctuations of hydrodynamic loads during its operation, especially when effected by surface waves, which may lead to blade damage due to fatigue. Therefore, the physical wave tank at the University of Galway is referenced and a numerical wave tank model is developed using the commercial Computational Fluid Dynamics (CFD) code ANSYS CFX with calibrated wave parameters from the wave tank as the input wave profile. This numerical wave tank can be combined with a previously developed tidal turbine model to evaluate the hydrodynamic performance of turbines operating in conditions influenced by ocean waves.

KEY WORDS: Computational Fluid Dynamics; Tidal Turbine; Wave; Fatigue; Tidal energy.

INTRODUCTION

As the Sustainable Development Goals (SDGs) developed by United Nations (2017) promote increasing substantially the share of renewable energy in the global energy mix, Marine and Renewable Energy Ireland (MaREl) Research Center forecasts that approximately 25 GW of renewable electricity capacity will be needed by 2050 compared to the 4.5 GW of capacity available in 2021 (MaREI & Wind Energy Ireland, 2021). Ocean energy, in the form of tidal energy, can play an important part in balancing Europe's electricity grid as it is predictable over large timescales. Tidal energy convertor technology has already attracted increased attention and investment worldwide due to its enormous potential with numerous highly energetic sites worldwide (O'Rourke et al., 2009).

However, during the nearly 20-year in-service design life, tidal turbines must be capable of contending with the harsh submarine conditions and the challenge of cyclic loads of tides and waves (Glennon et al., 2022), which poses a high demand for the long-term durability of turbine structures. Therefore, the loadings on the tidal turbine need to be accurately evaluated within the design stage, with little need for repair during service.

Many detailed representations of energy capturing system components of tidal turbines have been investigated in present studies, including hydrodynamic performance, blockage, supporting structures and device clusters, among which the fluctuation of loadings on tidal turbines attracts more and more attention recently as it may cause fatigue damage to turbine blades. For example, Tatum et al. (2016) found that in a pure tidal current situation, the load fluctuation on each blade is approximately 17%, while in a current/wave situation, the fluctuation could range from 32% and 36% of the mean thrust on each blade. Finnegan et al. (2020) found that the magnitude of fatigue loading on tidal turbine blades is very significant, which can be up to 43% of the maximum total thrust force on the blade. From the experimental dataset of the MaRINET2 project, it is found there exists notable fluctuations in the hydrodynamic loadings on the tidal turbine including thrust and torque force within the rotation of the turbine, especially on one blade under wave and current conditions (Xu et al., 2022). The results of Noruzi et al. (2015) showed that when the ratio of installation depth from the ocean surface to the total depth of water at the point of installation is smaller than 20%, the surface wave has a great role in the performance of the tidal turbine, and the uneven distribution of power and thrust spikes could lead to non-uniform loading and eventually fatigue. Therefore, it is essential to combining fatigue into the design of tidal turbine blades.

To understand the fluid-turbine interaction involved in operation, inevitably laboratory experiments have been conducted in water tanks, followed by scale-model tests in real sea conditions. However, with the recent advancement of computational capabilities, Computational Fluid Dynamics (CFD) offers a relatively inexpensive method of simulating the tidal turbine working environment and estimating loadings, compared to traditional physical testing, while maintaining accuracy and versatility under a range of operating conditions. To study the loading fluctuation on tidal turbines under wave conditions using CFD, developing a numerical wave tank model is the first step.

The numerical models for wave tank simulations are known as the numerical wave tank (NWT), in which waves are generated at the input boundary and damped out near the output boundary. A variety of numerical techniques have been used to develop NWTs, such as the boundary volume method (Yan & Liu, 2011), the finite element method (Turnbull et al., 2003) and the finite volume method (Finnegan & Goggins, 2012). Furthermore, commercial software packages, which are based on the Reynolds averaged Navier-Stokes equations, are widely used in present studies. For example, Liang et al. (2010) used ANSYS FLUENT to explore the use of a piston-type wavemaker to generate an irregular wave train using the finite volume method and compared the results to the results from an experimental wave tank. Lal and Elangovan (2008) used ANSYS CFX to conduct the CFD simulation of linear water waves for a flap-type wavemaker, however, the simulations were only carried out for the shallow water case.

In this paper, a CFD numerical wave tank model is developed using the commercial finite volume method package ANSYS CFX, capable of replicating measured waves from the wave tank at the University of Galway. For future work, this developed CFD wave tank model can be combined with the previously developed tidal turbine model by author (Xu et al., 2023) to investigate the hydrodynamic performance of a tidal turbine under wave conditions.

METHODOLOGY

Aim and objectives

The aim of this paper is to provide an overview of the methodology for developing a CFD wave tank model, with the wave parameters calibrated against data from a physical wave tank, which is located at the University of Galway. This model will be used to generate the wave input to investigate the hydrodynamic performance of a tidal turbine under wave conditions in the future. However, to achieve this aim, the following specific objectives must be completed:

- Calibrate the wave parameters using wave gauges in the wave tank at the University of Galway,
- Develop a CFD wave tank model with the calibrated wave parameters as wave input,
- Combine the developed wave tank model with the previously developed tidal turbine model for future study on the hydrodynamic performance of a tidal turbine under wave conditions.

Methodology overview

In this research, the physical wedged-shaped plunger-type wavemaker in the Hydraulic and Aerodynamics Laboratory at the University of Galway is referenced for model geometry and to provide calibrated wave parameters as the inflow profile in the numerical CFD wave tank. A numerical CFD tank model is developed using the commercial code ANSYS CFX, in which the inlet velocity profile method is adopted to replicate waves produced from the physical wave tank by setting up the wave elevation and water particle velocities of input waves before calculation. In addition, the Volume of Fluid (VOF) method is applied in the numerical wave tank model to determine the position of the free air-water boundary surface. Finally, a previously developed tidal turbine model is combined with the wave tank model to conduct hydrodynamic investigations for future reference.

PHYSICAL WAVE TANK

Experimental set-up

The wave tank at the Hydraulic and Aerodynamics Laboratory located in the Alice Perry Engineering building, University of Galway is utilised to provide reference data and model validation. The details of the wave tank are shown in Figs. 1~2. The wave tank has overall dimensions of 10 m x 1 m x 1 m. In order to generate waves, in the wave tank, there is a wedged-shaped plunger-type wavemaker in place, which is connected to a linear motor ensuring that it oscillates in a sinusoidal motion. This wavemaker system is capable of generating regular linear and non-linear waves with a wave period range of 0.5 to 2.1 s and the maximum stroke length is approximately 0.4 m. The total height of the wavemaker wedge is 0.6 m with the angle of the wedge as 26.5° and a plunger sectional area coefficient of 0.5. The wedge's base oscillates around a mean position of 0.35 m from the base of the flume and, therefore, the mean width of the plunger at the SWL is 0.175 m. Furthermore, the space between the wavemaker wedge and tank walls is 5 to 10 mm (Finnegan, 2013).

Fig. 1. Image of the wedged-shaped plunger-type wavemaker in the Hydraulic and Aerodynamics Laboratory at the University of Galway.

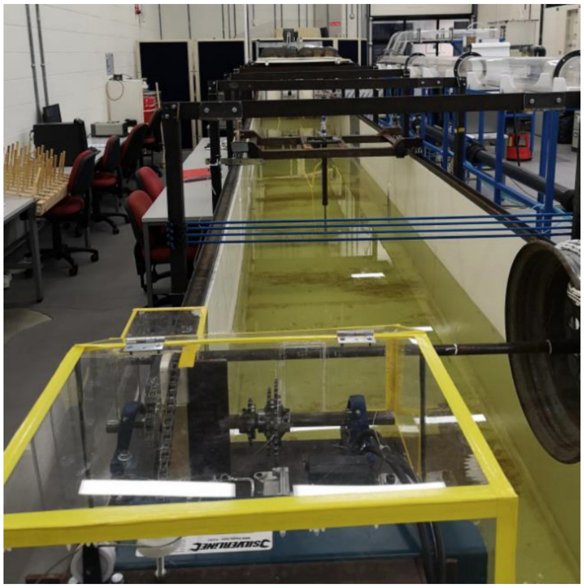


Fig. 2. Image of the wave tank with the towing rig, with the towing speed designed to simulate the inflow current velocity.

At the opposite end of the wave tank to the wavemaker, there exists a wave energy dissipation zone. To reduce the length of the wave energy dissipation zone and improve the energy dissipation efficiency, a combination of a varying slope beach and a wire mesh with sponges along the still water level is used in the lab wave tank. The beach is at a slope of 3:1, but a wedge is added to ensure the wire mesh with sponges remains along the still water level for 0.7 m to maximise the level of wave energy dissipation, as shown schematically in Fig. 3. Thus, the effective length of the wave tank is approximately 7 m.

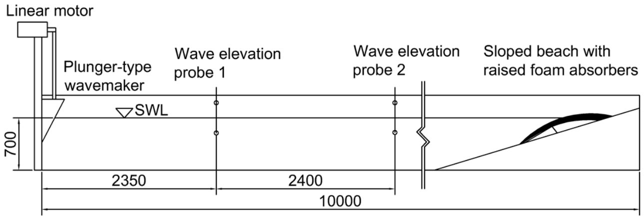


Fig. 3. Schematic of the experimental wave tank at the University of Galway (Including positioning of wave elevation probes). Dimensions are in mm (Finnegan, 2013).

Wave data calibration

A series of waves with different wave heights and wave periods were generated in the wave tank by adjusting the stroke length and motor voltage settings. The wave parameters were calibrated using the wave elevation probes. The representative calibrated wave results are presented in Fig. 4. These calibrated wave heights and wave periods are utilised as the wave input data for the development of the numerical CFD wave tank model.

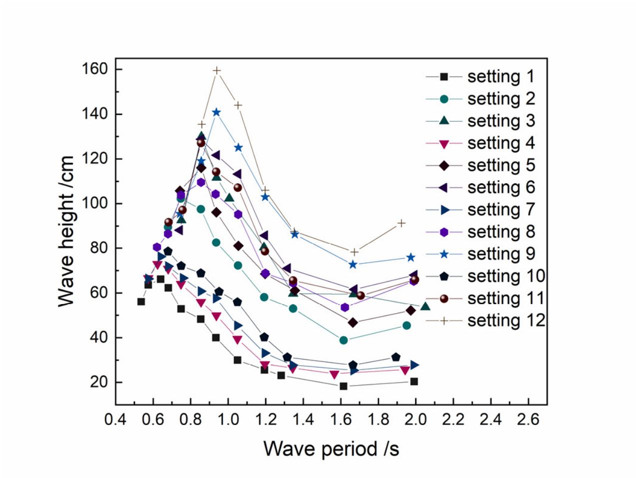


Fig. 4. Variation of calibrated wave heights with wave periods under a series of different wave stroke length and motor voltage settings in the wave tank at University of Galway.

Hydrodynamic test plan

The hydrodynamic testing of model turbines has been scheduled to be conducted in that wave tank at University of Galway to study on the hydrodynamic performance of tidal turbines under wave conditions and validate the numerical outputs from the developed wave tank model. The 3D printed scaled tidal turbine models have been prepared through the 3D printer service provided by University of Galway, as shown in Fig.5.

Fig. 5. 3D printed scaled tidal turbine models at the University of Galway.

NUMERICAL WAVE TANK MODELLING

As discussed previously, the fluctuation of hydrodynamic loadings on a turbine can be quite large with the addition of waves. Therefore, it is necessary to investigate the hydrodynamic performance of tidal turbines under wave conditions. Thus, a numerical CFD wave tank model needs to be developed first.

To replicate waves accurately at the testing location, one common method is to derive and input a wave profile with corresponding water particle velocities in the CFD model. In this wave tank model, the wave elevation and the water particle velocities of the input waves are set up before calculation. One of the major advantages of this inlet velocity profile method is that compared with using various types of wavemakers to generate waves, the computational time is greatly reduced as there are no moving boundaries. On the contrary, the wavemaker method requires a moving boundary and thus remeshing at each time step. The Volume of Fluid (VOF) method is adopted in this numerical wave tank model as well to determine the position of the free air-water boundary surface, as it provides an economical way to track free boundaries and is more efficient and flexible than the finite difference method in dealing with problems involving highly complex free surface configurations (Liu et al., 2021).

Geometry

In order to improve the computational efficiency of the developed wave tank model, a 3D geometry with a thickness smaller than the size of one meshing element is used in this model. Both the front and back boundaries of the modelled wave tank were set up as a symmetry boundary condition, so that the numerical wave tank model can be regarded infinitely wide. To be consistent with the wave tank at the University of Galway, which is 10 meters long and 1 meter high with a slope of 3:1 wave energy dissipation beach, the numerical model geometry was set up with the same parameters, pre-designed for the model validation and outcome comparison with hydrodynamic testing.

Mesh and pre-solver set-up

As the volume of fluid method uses the volume fraction to define the free surface water level, it is necessary to refine the mesh at the still water level (SWL) in order to improve the accuracy of the free surface. The mesh refinement technology utilised in this model is similar to that employed by Finnegan and Goggins (2012), i.e., the 'Sphere of Influence' mesh refinement method. Considering the thickness of the refined mesh around the SWL (0.7m high) needs to be larger than the maximum amplitude of waves (0.08m), it was set up as 0.2m above and below the SWL. The overall meshing was set up with a domain maximum element size of 0.05m and a refinement element size of 0.015m around the SWL, resulting in 73,107 elements (366,521 nodes) in total, as shown in Fig. 6.

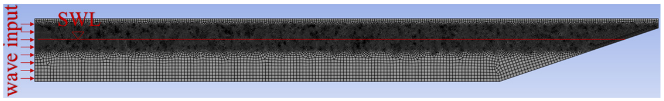


Fig. 6. Schematic of the meshing for the CFD wave tank model, the location of wave input boundary, the SWL and the mesh refinement along the SWL.

With the Volume of Fluid method, the surface tension at the air-water interface is assumed to be negligible. An initial hydrostatic pressure was set up and the entire region is static initially. The fluid temperature was set up to be 25°, therefore the density of air was specified to be 1.185kg/m^3 and the density of water was given as 997kg/m^3. The isothermal homogeneous heat transfer model was chosen. The top of the model was set up as an 'opening' boundary condition, which allows air to pass through. At the inflow boundary, the horizontal and vertical water particle velocities, as well as the wave elevation have been specified. The water inflow boundary type is also chosen as the 'opening' boundary condition, which allows fluid across the boundary in either direction. 'Opening' boundary is necessary at the water inflow boundary, as when specifying inflow velocities, which are specified both in and out of the fluid domain, 'opening' boundary allows any reflected water particles from inside the computational domain to pass through the boundary without being reflected back into the fluid domain, ensuring that the incoming wave will not be affected. The volume fraction is utilised in the boundary conditions to differentiate between the 'Water' velocities and 'Air' velocities. These inputs were set up using the ANSYS CFX expression language (CEL) (ANSYS CFX-Solver Theory Guide, 2021). At the outflow boundary, there is a hydrostatic pressure specified over the water depth using a CEL expression to initialise the SWL and allow for the overspill of excess water and the passing of air. As aforementioned, symmetry boundary conditions were specified for the adjacent sides. The remaining boundaries were assigned as static wall boundary conditions.

Wave input

The input wave of the CFD model was created using the ANSYS CFX Expression Language based on the airy wave theory, by manually setting up the expressions of the wave elevation and water particle velocities at the inflow and outflow boundary of the model. In the airy wave theory, the wave elevation can be obtained using Eq. 1:

$$\eta(x,t) = \frac{H}{2} cos\left(\frac{2\pi x}{L} - \frac{2\pi t}{T}\right) \tag{1}$$

where H is the wave height, x is the horizontal distance, L is the wavelength, t is the time and T is the wave period.

The wave height H and wave period T calibrated from the wave gauges in the wave tank are utilised. According to Airy wave theory, as the waves produced in the physical wave tank mainly belong to the deep and transitional water cases, the wavelength L, the water particle horizontal velocities u and vertical velocities w can be obtained using Eqs. 2~7:

Deep water ($\frac{d}{L} > \frac{1}{2}$):

$$L = \frac{gT^2}{2\pi} \tag{2}$$

$$u = \frac{\pi H}{T} e^{\frac{2\pi z}{L}} cos\left(\frac{2\pi x}{L} - \frac{2\pi t}{T}\right) \tag{3}$$

$$w = \frac{\pi H}{T} e^{\frac{2\pi z}{L}} sin\left(\frac{2\pi x}{L} - \frac{2\pi t}{T}\right) \tag{4}$$

Transitional water ($\frac{1}{25} < \frac{d}{L} < \frac{1}{2}$):

$$L = \frac{gT^2}{2\pi} \tanh \frac{2\pi d}{L} \tag{5}$$

$$u = \frac{H}{2} \frac{gT}{L} \frac{\cosh[2\pi(z+d)/L]}{\cosh(2\pi d/L)} cos\left(\frac{2\pi x}{L} - \frac{2\pi t}{T}\right) \tag{6}$$

$$w = \frac{H}{2} \frac{gT}{L} \frac{\sinh[2\pi(z+d)/L]}{\cosh(2\pi d/L)} sin\left(\frac{2\pi x}{L} - \frac{2\pi t}{T}\right) \tag{7}$$

where d is the water depth, g is the acceleration of gravity.

RESULTS

The aforementioned calibrated wave data from the numerical wave tank at the University of Galway are utilised to generate a series of waves with different wave-particle inputs in the wave tank model through the ANSYS CFX Expression Language. For example, in the case of a wave height of 0.13m and a wave period of 0.86s, the free surface of the model after calculation is shown in Fig. 7 for a specific moment in time.

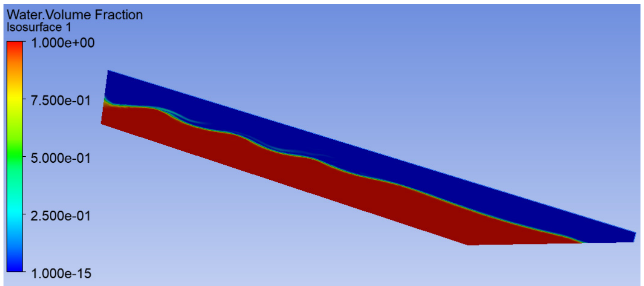


Fig. 7. Schematic of the free surface and simulated waves from CFD analysis after 4.085s.

Comparison

The wave outcome from the numerical wave tank model is compared against the experimentally calibrated wave data from the physical wave tank within one wave period, as shown in Fig. 8. Though there exists some deviations, the overall wave amplitude and wave period are quite similar, validating the accuracy of the numerical wave tank model.

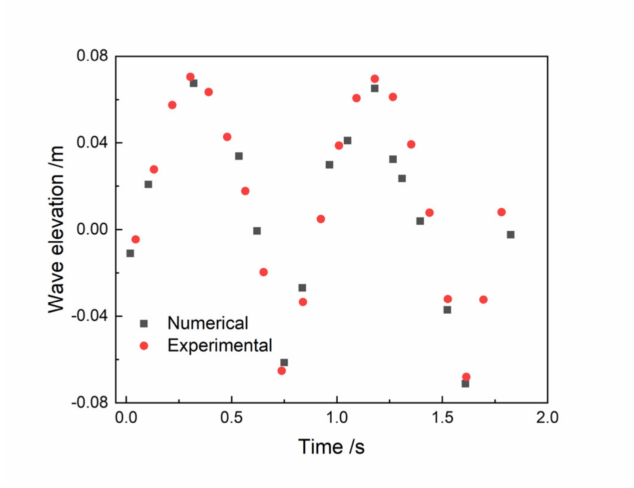


Fig. 8. Comparison of wave outcome from the numerical wave tank model against the experimentally calibrated wave data from the physical wave tank within one wave period.

CONCLUSION

Ocean energy, in the form of tidal energy, can play an important part in balancing Europe's electricity grid considering its enormous potential with numerous highly energetic sites and its advantages, in terms of reliability and predictability. The loadings on tidal turbine blades need to be evaluated accurately within the design stage to ensure their durability with little need for repair as they need to withstand harsh marine conditions during the nearly 20 years in-service life.

In this paper, a numerical wave tank model is developed, which shares the same size as the physical wave tank at the University of Galway, in consideration of model validation and results comparison. The experimentally calibrated wave parameters in terms of wave height and wave period from the wave tank at the University of Galway are utilised as wave input to generate linear waves based on Airy Wave Theory with the inlet velocity profile method. Furthermore, a CFD analysis of the hydrodynamic performance of a tidal turbine under wave conditions has been prepared by combining the numerical wave tank and formerly developed 3D tidal turbine model together.

FUTURE WORK

The thickness of the numerical wave tank can be extended to 1m wide, just the width of the physical wave tank at the University of Galway. The boundary conditions of the front and back sides of the wave tank change to static walls correspondingly. For the same case of a wave height of 0.13m and a wave period of 0.86s, the free surface of the calculated numerical wave tank model is shown in Fig. 9 for a specific moment in time.

The next step of the research will be to conduct the simulation of the developed wave tank model combined with the tidal turbine model to study the hydrodynamic performance of the tidal turbines under wave conditions. Some initial meshing and pre-solver setups have been carried out, as shown in Figs.10~11.

The fluctuation of hydrodynamic loadings on turbine blades will be systematically researched as it may lead to turbine blade damage due to fatigue. Meanwhile, the hydrodynamic testing of 3D printed scaled turbine models is planned and scheduled in the wave tank at University of Galway for numerical model validation and to study on the fluid-structure interaction between the turbine and waves, in order to gain a greater understanding of the operating performance and fatigue damage mechanisms of tidal turbines blades due to ocean waves.

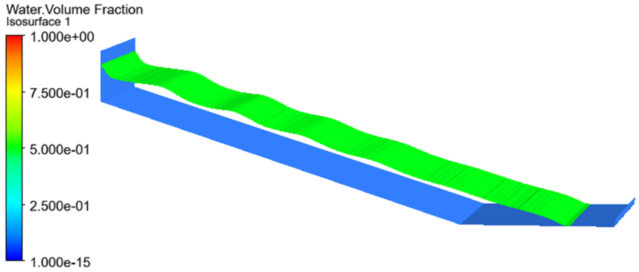
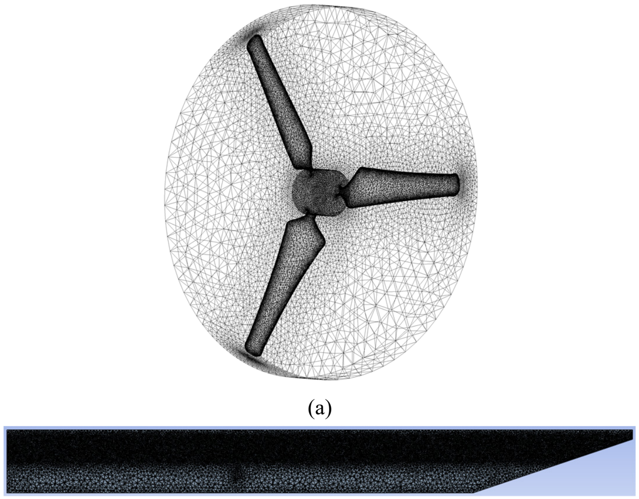


Fig. 9. Schematic of the free surface and simulated waves of the numerical wave tank from CFD analysis after 6.751s.



(a)

(b)

Fig. 10. Schematic of the meshing for combined wave tank with a 3D tidal turbine model: (a) rotating domain; and (b) stationary domain.

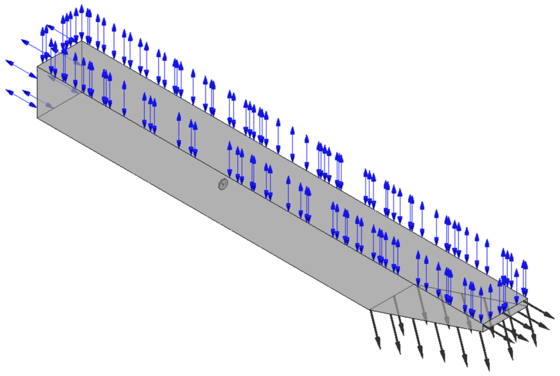


Fig. 11. Schematic of the pre solver setup for combined wave tank with a 3D tidal turbine model.

ACKNOWLEDGEMENTS

This research was funded in part by Science Foundation Ireland (SFI) through the MaREI Research Centre for Energy, Climate and Marine (Grant no. 12/RC/2302_2) and the European Commission through the H2020 CRIMSON project (grant agreement no.: 971209).

REFERENCES

ANSYS (2021). *ANSYS CFX-Solver Theory Guide*.

Finnegan, W (2013). *Wave-structure interaction of offshore wave energy converters*.

Finnegan, W, Fagan, E, Flanagan, T, Doyle, A, and Goggins, J (2020). Operational fatigue loading on tidal turbine blades using computational fluid dynamics. *Renewable Energy*, 152, 430–440.

Finnegan, W, and Goggins, J (2012). Numerical simulation of linear water waves and wave-structure interaction. *Ocean Engineering*, 43, 23–31.

Glennon, C, Finnegan, W, Kaufmann, N, Meier, P, Jiang, Y, Starzmann, R, and Goggins, J (2022). Tidal stream to mainstream: mechanical testing of composite tidal stream blades to de-risk operational design life. *Journal of Ocean Engineering and Marine Energy*.

Lal, A, and Muniyandy Elangovan (2008). CFD simulation and validation of flap type wave-maker. *World Academy of Science, Engineering and Technology*, 46, 76–82.

Liang, XF, Yang, JM, Li, J, Xiao, LF, and Li, X (2010). Numerical simulation of irregular wave-simulating irregular wave train. *Journal of Hydrodynamics*, 22(4), 537–545.

Liu, J, Xu, Y, Zhou, K, and Ge, MF (2021). Fault diagnosis and failure prognosis in hydraulic systems. *Fault Diagnosis and Prognosis Techniques for Complex Engineering Systems*, 207–262.

MaREI, and Wind Energy Ireland (2021). *Our Climate Neutral Future: Zeroby50*.

Noruzi, R, Vahidzadeh, M, and Riasi, A (2015). Design, analysis and predicting hydrokinetic performance of a horizontal marine current axial turbine by consideration of turbine installation depth. *Ocean Engineering*, 108, 789–798.

O'Rourke, F, Boyle, F, and Reynolds, A (2009). Tidal energy update 2009. *Renewable Energy*, 3, 451–476.

Tatum, S, Allmark, M, Frost, C, O'Doherty, D, Mason-Jones, A, and O'Doherty, T (2016). CFD modelling of a tidal stream turbine subjected to profiled flow and surface gravity waves. *International Journal of Marine Energy*, 15, 156–174.

Turnbull, MS, Borthwick, AGL, and Eatock Taylor, R (2003). Wave-structure interaction using coupled structured-unstructured finite element meshes. *Applied Ocean Research*, 25(2), 63–77.

United Nations (2017). Global indicator framework for the sustainable development goals and targets of the 2030 agenda for sustainable development Indicators. In *United Nations Statistics Division*.

Xu, K, Finnegan, W, O'Rourke, F, and Goggins, J (2022). Hydrodynamic modelling of marine tidal turbines: A state of the art review. *Civil Engineering Research in Ireland Conference (CERI 2022)*, 520–524.

Xu, K, Finnegan, W, O'Rourke, F, and Goggins, J (2023). CFD analysis of hydrodynamic force on a horizontal axis tidal turbine. *Proceedings of the European Wave and Tidal Energy Conference*, 15.

Yan, H, and Liu, Y (2011). An efficient high-order boundary element method for nonlinear wave-wave and wave-body interactions. *Journal of Computational Physics*, 230(2), 402–424.