journal homepage: www.brodogradnja.fsb.hr

Brodogradnja

An International Journal of Naval Architecture and Ocean Engineering for Research and Development

Experimental and numerical investigations of hydrodynamic performance for horizontal-axis hydrokinetic turbines



BRODOGRADNJA

63

Fatih Mehmet Kale^{1*}, Naz Yilmaz¹, Kemal Furkan Sokmen², Weichao Shi³

¹Department of Naval Architecture and Marine Engineering, Maritime Faculty, Bursa Technical University, Bursa, Turkiye ²Department of Mechanical Engineering, Faculty of Engineering and Natural Sciences, Bursa Technical University, Bursa, Turkiye

³School of Engineering, Newcastle University, Newcastle Upon Tyne, United Kingdom

ARTICLE INFO

ilty of Mechanical Engineer

Naval Architecture

Keywords: Horizontal-axis hydrokinetic turbines (HAHT) Hydrodynamic performance Experimental investigation Computational fluid dynamics (CFD) Power coefficient (*C*_P) Thrust coefficient (*C*_T)

ABSTRACT

This paper presents both experimental and numerical investigations of the hydrodynamic performance of Horizontal-axis Hydrokinetic Turbines (HAHTs) using experimental methods and Computational Fluid Dynamics (CFD) approaches, respectively. The innovative aspect of this study lies in the consistency of the results, achieved by aligning the method used in the CFD analyses for Hydrokinetic Turbines (HKTs) and airfoil profiles with experimental data. For this purpose, 2-D CFD analyses were first conducted with blade section geometries (Eppler 395 and S1210), which are commonly used in HKT designs. The aerodynamic characteristics (C_L and C_L/C_D) of these blade sections were computed and compared with the experiments. Subsequently, a three-dimensional (3-D) turbine geometry, featuring three different pitch angles (PAs), was simulated using CFD, and the results were compared with experimental data obtained under the same operating conditions in the Emerson Cavitation Tunnel (ECT) at Newcastle University. The comparisons showed good agreement while the maximum relative error was calculated less than 10 % for the power coefficient (C_P) of the turbine with a PA of 0°. For the other PA (8°), the maximum relative error was 11 % for C_P and 14 % for the thrust coefficient (C_T). The CFD investigations of HKTs revealed that the Detached Eddy Simulation (DES) model has less relative errors compared to the other turbulence models at the same Tip Speed Ratio (TSR) values, while the Sliding Mesh (SM) method describing rotation gives more consistent and closer results to the experiments, with the investigation of y^+ point of view.

1. Introduction

In the context of contemporary environmental challenges, the importance of renewable energy is increasing. The main reason for this is the formation of emissions that pollute the environment as a result of energy obtained from fossil fuels. A key area of contemporary research involves increasing the performance and efficiency of systems working with renewable energy to obtain clean energy yield. The main renewable

* Corresponding author.

E-mail address: fatihmehmet.kale@btu.edu.tr

http://dx.doi.org/10.21278/brod76308

Received 10 February 2025; Received in revised form 26 May 2025; Accepted 30 May 2025 Available online 4 June 2025 ISSN 0007-215X; eISSN 1845-5859

energy sources are sunlight, wind, wave and tidal. Solar panels, wind turbines and HKT are used to convert renewable energy into electrical energy [1, 2].

The design purpose of HKT is to convert the kinetic energy of the flowing water into usable electrical energy. HKTs have attracted increasing attention in recent years because they can use energy-promising sources such as tides, ocean waves, and rivers with minimal environmental impact unlike dams, which can lead to significant ecological issues.

Many research studies were examined about the predictions of the hydrodynamic performances of the HKTs using numerical and experimental approaches during this research work.

HKTs are similar to scaled-down wind turbines or have axial flow based on the Darrieus cross-flow principle [3]. According to the energy conversion technique, hydrokinetic energy converters are classified as turbine type and non-turbine type [4, 5]. The designs of most HKTs are conceptualized from wind turbine rotors, and then design modifications are made to suit the operating environment. Many research studies show that the performance of HKT is superior to that of wind turbines [6, 7]. The maximum power obtained by the HKT operating under the same operating condition was compared with the wind turbine [7].

Primarily, HKTs are designed based on drag and lift techniques. While the Savonius HKTs are based on drag, Darrieus and Gorlov HKTs are operating based on lift. HAHT which are the main interest of this study, have higher efficiency (C_P) compared to Vertical axis Hidrokinetic Turbines (VAHTs) [5].

Before the 3-D CFD investigations of HKTs, the most used sections for HKT design were examined for two-dimensional (2-D) validation purposes. The various airfoils and hydrofoils are used in many turbine blade designs today. Researchers either used existing foils for turbine blade designs or designed turbine blades with newly designed foils. They also used foils for the diffuser and stator designs to increase the hydrodynamic performance of the turbines. When choosing airfoils, researchers made their choices by defining optimum Angles of Attack (AoA), lift coefficient (C_L), drag coefficient (C_D), and C_L/C_D ratios or pressure distributions and by comparing their hydrodynamic and aerodynamic performances among themselves. This study initially conducted 2-D CFD analyses with blade section geometries (Eppler 395 and S1210), which are commonly used in HKT designs. The researchers developed a wind turbine blade using the S1210 blade cross-sectional geometry [8]. In another study, the blade cross-sectional geometries in which the S1210 blade cross-sectional geometry is included were simulated using CFD. The purpose of doing this is to determine the optimal blade cross-sectional geometry for low-speed Fixed-Wing UAV [9]. Another research group also tried to estimate aerodynamic performance of NACA0018 blade cross-sectional geometry with CFD using the ANSYS Fluent [10]. The researchers made numerical estimates of the cavitated and non-cavitated state of the flow around the 2-D NACA66MOD hydrofoil using the standard k- ε turbulence model [11].

Upon reviewing previous research studies on HKTs, it has been concluded that HKTs are generally divided into two categories regarding their working and flow axis: horizontal and vertical ones. The researchers applied deflector designs to increase the C_P values of VAHT. It is observed that an increase in the C_P is achieved at low velocity, but in related studies, it is observed that the C_P cannot exceed the value of 0.3 [12-14]. When we look at the studies on HAHTs, the power coefficient at low velocity is around 0.45. Without the use of a support element that will provide any power increase [15-17].

The purpose of choosing HAHTs in this study is that they are better designs in terms of hydrodynamic efficiency and performance compared to others [5]. Another research group compared the efficiency and rotor performance of two HAHTs, designed as a multi-element hydrofoil with an airfoil and a classical design hydrofoil, experimentally and numerically. They examined C_P values with TSR varying between 2.5 and 9.0 [18]. The vortex characteristics of the HAHT were investigated at various TSR using velocity measurements and flow visualization techniques. As a result of the experimental results, it was shown that the vortex structures formed in the lower region of the turbine up to six rotor diameters are significantly affected by changes in the TSR [19]. The research studies of HAHTs involving vortices were reviewed, data on the vortex propagation length were obtained, and the possibility of developing an equation to predict the vortex propagation length using the collected data was evaluated [20]. A preliminary HKT prototype was designed for river applications. A circulation water channel was built to measure the C_P values of this HKT. The values found as a result of the experimental study were verified by numerical method [21]. The performance of

coaxial HAHT mounted on a single shaft was evaluated. In the experimental study, the C_P at different TSR was compared with the performance of a 3-bladed turbine, the performance of two sequential 3-bladed turbines, and the performance of a 6-bladed turbine. The hydrodynamic performance of blades made of carbon fibre polymer composites was evaluated in a water tunnel [22]. The study of HAHT revealed that the tip vortex breaking mechanism depends on the TSR. For lower TSR values, instabilities in the root vortex core were found to cause distortion. The results obtained in different experiments of the HAHT have been numerically verified. It aimed to improve the hydrodynamic performance of the HAHT to get as close as possible to the theoretical energy conversion efficiency limitation of 59.3 %, known as the Lanchester-Betz-Joukowsky limit. Fins were integrated into the turbine blade tips to reduce the effect of tip vortex. The aileron design concept was inspired by jet aircraft and rearranged for the HKT application [23].

Building on our research into HAHT studies, one of the priorities in choosing HKTs is the use of renewable energy sources and obtaining power through kinetic energy from HKTs, which is one of these usage methods. Since this method of obtaining power is more environmentally friendly and the continuity of renewable energy sources found in nature is in question, the importance of HKTs is increasing. As a result, numerous studies have been conducted in response to these increasing demands. In these studies, the important factors were examined by numerical or experimental methods in terms of both power and efficiency increment. Considering the studies in literature, flow analyses of HKTs are performed in commercial software such as Ansys Fluent and Star CCM+ and open-source codes like Open FOAM. The RANS *k*-ω Shear Stress Transport (SST) model is mainly used as modelling turbulence for the analysis of the flow around the turbine. However, although DES and other RANS models are widely used in CFD computations, they seem to be rare in HKT studies. In the same way, the Moving Reference Frame (MRF) method is used as the rotation method in the studies usually performed, while the SM method is used more rarely than the MRF method due to the complex modelling and computational time. When the studies in literature are examined, it is seen that DES method is rarely used as a turbulence model, but it gives more consistent results in terms of C_P and C_T when experimental result comparisons are made in HKT studies [23]. At the same time, it is observed that SM method gives closer results to the experiments compared to MRF method when it is used as a rotation method [24], this study was conducted to determine which solution method was used, especially in numerical analysis, or which turbulence model gave closer results to the experimental results, or which rotation method was suitable for describing experimental conditions properly. Many different HKT designs were used to carry out these investigations. For this reason, they used different methods for the design of HKT blades. While designing these blades, blade section profiles and PA which are the most important variables and are still investigated today, were taken into consideration for this research study.

Following the introduction part above about HKTs and HAHTs, this manuscript continues with the numerical and experimental investigations for 2-D blade section profiles, which are the most used for HKT designs and the experimental investigation of the hydrodynamic performance of 3-D HKT design is presented in Section 2, the numerical computations are demonstrated in Section 3. Section 4 provides a comprehensive analysis of the effects of turbulence models, rotation methods, and mesh density and quality. Finally, the paper concludes with a discussion of the findings and outlines future work in Section 5.

2. Methodology

2.1 Methodology description

The hydrodynamic and aerodynamic performances of HKT and foil profiles were investigated using the CFD method. Firstly, the foil profiles used in HKT designs were investigated. Based on this analysis, S1210 and Eppler 395 foil profiles were selected. The aerodynamic performance of the selected Foil profiles in $Re=2x10^5$ and $Re=1.65x10^5$ was verified with CFD and experimental results. Later, the experimental study of the HKT was carried out. In order to predict the results of the experimental study and the hydrodynamic performance of the HKT, CFD was set up. In the CFD verification study of the hydrodynamic performance of a HKT, turbulence model comparison was performed first. Then, the rotation method, mesh density and y^+ effect were examined. As a result of the CFD verification study, the methods that can predict the performance

of the HKT most consistently were selected and compared with the experimental result in all TSR values in terms of C_P and C_T . Figure 1 shows the methodology workflow diagram.



Fig. 1 Methodology description workflow diagram

2.2 Blade section profiles for HKTs (2-D)

In a previous study, the researchers investigated the foil profiles used in HKT designs. As a result of the research, the C_L and C_L/C_D values of the profiles used were compared. This comparison was made with the Reynolds numbers (*Re*) 2x10⁵ value. As a result of the comparison, foils with the highest C_L value in the 5 and 10° AoA were listed as follows: Eppler 423, S1210, Wortmann FX 63-137, and Eppler 395. In the same way, the ranking of C_L/C_D values is Wortmann FX 63-137, Eppler 395, S1210, and NACA 4412 [25].



Fig. 2 The computational domain and mesh

For validation purposes and to investigate the performance parameters of the blade section profiles for HKT designs, two different section profiles were selected. These are the Eppler 395 and S1210 sections. The purposes of selecting these profiles are presented as follows:

- The airfoil of the Eppler 395 has quite high efficiency (C_L/C_D) according to XFOIL data [25].
- The C_L values of the S1210 section profile are much higher compared to other blade cross-section profiles [25].

Since HAHTs operate based on C_L and efficiency is crucial for all turbines, these two airfoils were preferred within the scope of these analysis studies. The analyses were performed using the CFD approaches in ANSYS Fluent (Academic).

Firstly, in the analyses, the geometric coordinates are needed to create the selected Eppler 395 and S1210 airfoils. The 2-D coordinates of the selected airfoils are taken from the Airfoil Tools. The domain was prepared in the same dimensions for the Eppler 395 and S1210 airfoils. The chord length (C) of the geometries used in the analyses is 0.1 m. The computational domain was modelled concerning the 2-D airfoil verification studies that were previously conducted and published in the open literature [26]. The dimensions of the domain and the boundary conditions used in the analyses (velocity inlet, non-slip wall, and pressure outlet) are shown in Figure 2.

	C_L	C_D
N_{I}	1660416	1660416
N_2	755846	755846
N3	345215	345215
<i>r</i> ₂₁	1.4821	1.4821
r32	1.4797	1.4797
	1.1682	0.02924
	1.16664	0.02905
Ø3	1.16549	0.02844
821	-0.00156	-0.00019
E32	-0.00115	-0.00062
S	1	1
<i>e</i> _{a21}	0.001333663	0.00644
q	0.00488	0.00723
p_a	0.75204	3.04134
	1.1727	0.0293
Cext21	0.00386	0.00278
GCI _{fine21}	0.00484	0.00348
GCIasymptotic	1.00967	0.91311

Table 1 Mesh independecy with GCI method for Eppler 395

The mesh structure and the total number of cells were generated the same for the Eppler 395 and S1210 airfoils. The analyses were performed in 3 different total cell amounts, which were coarse, medium and fine. The total number of cells (*N*) is 345K, 755K and 1.66M, for coarse, medium and fine mesh densities, respectively. Particular attention was paid to the mesh quality to capture the curvature at the leading and trailing edges, and fine mesh was generated accordingly. For the Eppler 395 airfoil analyses, the medium mesh was preferred by taking into account the C_L and C_D values in the analyses performed under $Re=2x10^5$ and AoA (7.25°) conditions. Within the scope of the mesh independence study, The Grid Convergence Method (GCI) was applied [27]. The results of the GCI method were found to be $GCI_{fine21}=0.0048$ and $R_{fine}=0.74$ for C_L values. For C_D , $GCI_{fine21}=0.0034$ and $R_{fine}=3.28$ were found. The results of the GCI method are shown in Table 1. The total thickness value for the boundary layer was kept as minimum as possible so that the y^+ value is below 1 on the geometries used. The boundary layers have 12 layers with a growth rate of 1.2. The generated mesh structure for the boundary layer and on the blade section is also shown in Figure 2.

The analyses were established as pressure-based and time-independent (steady). The flow around the blade section was solved using the RANS method. The $k-\omega$ SST turbulence model were preferred due to better

prediction of the flow separations that will occur at the trailing edge of the blade geometry. In the relevant experimental study, the turbulence intensity was taken into consideration as 3 % [20]. The CFD analyses were conducted with this value for the relevant airfoil profiles. The air was defined as the fluid for validation purposes in the analyses performed. The velocity-inlet for the inlet patch and pressure-outlet for the outlet patch were described as boundary conditions. A SIMPLE algorithm and 2^{nd} order upwind are used for the solution. C_L and C_D are defined in the reporting section so that they can be followed throughout the analysis until the convergence. The same simulation methodology was applied for both selected geometries.

The analyses were performed with $Re=2x10^5$ and $1x10^5$ values. In these Re, the aerodynamic coefficients with XFOIL and experimental results [20] were compared with the results of the analyses performed by the CFD approach. In the experimental study, while the uncertainty analysis result was generally found to be 1.5 % for the C_L , it was found below 1 % for the C_D [28]. Figure 3 presents the C_L and C_D values at different AoA of the analyses performed at the $Re=2x10^5$.



Fig. 3 C_L and C_D values at different angles of attack (*Re*=2x10⁵)

The Eppler 395 airfoil analyses were also performed using the same CFD methodology in the ANSYS Fluent. The analyses were conducted at $Re=2x10^5$ and $Re=1.65x10^5$ values at different AoA. The aerodynamic coefficient values were compared with the XFOIL and CFD values [29] had already been conducted before. To author's knowledge, no experimental or CFD studies in the literature have provided comparable results for the Eppler 395 airfoil at $Re=2\times10^5$. In the related study, due to the lack of experimental data, CFD analyses were performed at $Re=1.65x10^5$, and the aerodynamic coefficients of the airfoil were compared with the XFOIL data, but the relative error rates were found to be high [29]. Figure 4 demonstrates the C_L and C_D values at different AoA of the CFD analyses performed for the Eppler 395 ($Re=1.65x10^5$).



Fig. 5 C_L and C_D values at different angles of attack (*Re*=2x10⁵)

Figures 4 and 5, inspired by the study conducted [29], tried to estimate the values of C_L and C_D at $Re=2x10^5$ and these values were compared with XFOIL. As in other analyses, the C_L values were significantly different from the XFOIL predictions, while the C_D values showed relatively closer agreement. When the CFD analyses at $Re=1.65x10^5$ performed [29] and the CFD results of this study were compared with each other, the C_L values could be estimated with a maximum relative error rate of 5 % and the C_D values with a maximum relative error rate of 12 %.

When Figures 3-5 are examined, there appears to be declivity that decreases and increases in the C_L between 9 and 12° of AoA. After 12°, there is a decrease in the C_L , except for the S1210 foil profile. The analysis suggested that it reaches the stall angle at 12° for both foil profiles. The reason for this decrease in the C_L and the reason it does not increase anymore are the vortices formed in the trailing edge part of the foil profile. The vortices formed the trailing edge the foil profiles in part of are seen in Figures 6-8.



Fig. 6 S1210 (at stall angle 11.61° and $Re=2x10^{5}$)



Fig. 7 Eppler 395 (at stall angle 12° and $Re=2x10^{5}$)



Fig. 8 Eppler 395 (at stall angle 12° and $Re=1.65 \times 10^{5}$)

Looking at the vortex sizes formed on the Eppler 395 and S1210 foil profiles at the same *Re* number and close AoA, it is observed that a larger vortex is formed on the Eppler 395, as shown in Figures 6 and 7. This situation directly affects the C_L and C_D . While the S1210 foil profile provides more C_L , the Eppler 395 produces more C_D value. When Figures 7 and 8 are examined, it is seen that the size of the vortex formed increases as the *Re* number increases.

2.3 Experimental investigations for HKTs (3-D)

After the 2-D aerodynamic performance predictions of the mostly used airfoil sections for the HKT designs, the study aimed to continue with 3-D hydrodynamic investigations of a target HAHT model with experimental and computational fluid dynamics methods. For these investigations, a turbine model geometry that had been previously tested and simulated by several researchers was selected. Firstly, this HAHT was modelled and tested [30]. After that, existing experimental data were compared with CFD analysis results and the comparisons showed good agreement [31, 32].

2.3.1 Turbine geometry

The HKT geometry (Figure 4) was manufactured by Centrum Techniki Okretowej S.A. (CTO, Gdansk). Three pitch-adjustable (0° , 4° , and 8°) turbine geometries were tested in the ECT and were simulated for this research using CFD approaches in ANSYS Fluent. The radial chord (*C*) and pitch distribution for the blades were presented in Table 2, while the turbine diameter (*D*) was 400 mm for the model geometry. The S814 airfoil section was selected as the main blade section, which is also shown in Figure 9.



Fig. 9 The HKT geometry (model scale) and S814 airfoil [33]

r/R (-)			0.2	0.3	0.4	0.5	0.6	0.7	0.8	0.9	1
Chord (mm)	Length	(<i>C</i>)	64.35	60.06	55.76	51.47	47.18	42.88	38.59	34.29	30
Pitch A1	ngle (°)		27	15	7.5	4	2	0.5	-0.4	-1.3	-2

 Table 2
 Technical specifications of the turbine model

2.3.2 Experimental setup

The HAHT model was tested in the ECT at Newcastle University. While the sketch of the tunnel is shown in Figure 10, more details about the tunnel, which is a medium size cavitation tunnel with measuring section of 1219 mm x 806 mm, provided in [34]. The turbine was mounted on a dynamometer (K&R H33) for the measurement of the thrust and torque values of marine propellers or turbines. Technical information of the H33 dynamometer, nominal maximum thrust is ± 3000 N, nominal maximum torque is ± 150 Nm, and the maximum rotation speed is 4000 RPM. There is a 64 kW DC motor on the dynamometer to control the rotation speed. In the ECT, Particle Image Velocimetry (PIV) and Laser Doppler Anemometry (LDA)/Phase Doppler Anemometry (PDA) flow measurement systems are located in order to observe the cavitation occurring around the HKT. More information about the system used can be found in detail [34]. The uncertainty analysis was performed in the experiment, and the tests were repeated 3 times for each TSR value at a velocity (*V*) of 2 m/s. As a result, an average deviation of approximately 2.9 % in the *C*_P and 0.7 in the *C*_T was calculated for a HKT with an 8° pitch angle [33].



Fig. 10 The sketch of ECT, Newcastle University

3. Numerical investigations for HKTs (3-D)

3.1 Computational domain

The computational domain was divided into two parts: rotating and stationary. These are the area of the zone in which the rotor model rotates and an external region surrounding this region (Figure 11). The dimensions for the domain were used as in Figure 11 [35]. The flow field and its conditions are formed as follows: the patch where the flow enters the domain is defined as velocity-inlet, the part where it exits is defined as pressure-outlet, the outer region of the flow field and rotor geometry is defined as the wall.



Fig. 11 Computational domain for the rotor

3.2 Mesh generation with mesh independency study

The mesh structure generated in the analyses is irregular tetrahedral mesh (Figure 12). 10 layers were created on the rotor surface for the boundary layer thickness to keep y^+ value at a reasonable level. The y^+ value is generally tried to be kept between less than 5 or above 50, depending on the turbulence model used [18, 36]. In this study, y^+ study was also carried out for the investigation of the effects on hydrodynamic performance. For the mesh independency study, while the mesh was generating, GCI method was used [19]. The total number of cells (*N*) was 6.6 M, 8.6 M, and 12.6 M cells for coarse, medium, and fine meshes, respectively. Figure 12 demonstrates a fine mesh with perspective and side view. Regarding mesh quality, the maximum aspect ratio does not exceed 84, the maximum skewness was below 0.90, and the minimum

orthogonal quality was higher than 0.1 for each mesh. GCI method results were found to be $GCI_{fine21}=0.054$ and $R_{fine}=0.744$ for C_P values. For C_T values, $GCI_{fine21}=0.02$ and $R_{fine}=2.67$ were calculated. GCI_{fine} represents the uncertainty value of the numerical simulations in the GCI method, and if R_{fine} is found positive, it shows the convergence in the analysis (Table 3).



Fig. 12 Generated mesh (top - perspective, bottom - side view)

	C_P	C_T
N_I	12642282	12642282
N_2	8656507	8656507
N_3	6608082	6608082
<i>r</i> ₂₁	1.134560335	1.134560335
<i>r</i> ₃₂	1.094180793	1.094180793
	0.390433447	0.993323277
	0.371300414	0.976781176
Ø3	0.345590402	0.970602801
E21	- 0.019133033	-0.0165421
E32	-0.025710013	-0.00617838
S	1	1
e_{a21}	0.049004594	0.016653291
q	0.458050813	0.425086618
p_a	5.96866	4.43396
Ø _{ext21}	0.407448839	1.015371643
e _{ext21}	0.041760806	0.021714577
GCI _{fine21}	0.054475968	0.027745708
GCIasymptotic	2.67183	7.63137

Table 3 Mesh independency with GCI method

3.3 Numerical setup

The commercial CFD package programme, Ansys Fluent (Academic) Finite Volume Method (FVM) solver, was used in the present study to solve the governing equations such as momentum, continuity, etc. [37].

In CFD computations, fluid flows are computed using various methodologies depending on the nature of the flow and the capacity and performance of the computational resources. Numerical models regarding turbulence can be commonly categorized as RANS, Scale Resolving solvers DES and Large Eddy Simulation (LES), and Direct Numerical Simulation (DNS), which is the complex one. While the RANS approaches are generally used for the predictions of the rotating objects (propellers, fans, and turbines), scale resolving simulations are required for calculating turbulent flows with detailed investigations (separations, vortices, etc.).

For the CFD investigations of the HKTs, RANS (k- ω SST and Realizable k- ε) models and DES have been commonly used as turbulence models [21, 23]. In this study, different models were used and compared each other in terms of the hydrodynamic performance of the rotor model at the same operating condition (*@ TSR* 3) with the same mesh structure.

Two different rotation methods (MRF and SM (Mesh Motion)), which are mostly used models [24, 38] for the CFD investigations for the rotating objects were simulated and the effects on the turbine performance were investigated.

The turbine analysis was conducted as transient with time step values based on recommendations by The International Towing Tank Conference (ITTC) and others in the open literature. Hence, the time step was calculated so that the rotor rotates between 0.5° and 2° per time step (ITTC, 2014). The time step value corresponding to 1° of the rotor rotation was used for the simulations, and the results were obtained after at least 4 rotations of the turbine. For each TSR value, the angle scanned by the turbine blade was calculated accordingly. For each case, 1500 time steps were run using calculated time steps with 10 inner iterations. For the solution algorithm, the second order upwind SIMPLE was used [18].

4. Results

The hydrodynamic performance results of the turbine model were investigated with not only quantitative data (C_P and C_T) but also visual results such as pressure, velocity contours with streamlines, and isosurface with Q-criterion data in model scale for this part. The hydrodynamic performance parameters of the turbine; C_P and C_T were calculated using Equations (1) and (2), respectively, as follows.

$$C_P = \frac{Q \times \omega}{\frac{1}{2} \times \rho \times A_T \times V^3} \tag{1}$$

$$C_T = \frac{T}{\frac{1}{2} \times \rho \times A_T \times V^2} \tag{2}$$

where Q is the moment, ω is the angular velocity, ρ is the density of the fluid, A_T is the swept area, V is the velocity and T is the thrust.

The relative error between experimental and CFD results was calculated according to Equations (3) and (4) [39].

$$e_r(C_{P,T}) = \frac{|C_{P,T}^{CFD} - C_{P,T}^{EXP}|}{C_{P,T}^{EXP}} x100$$
(3)

$$e_r(C_{L,D}) = \frac{\left|C_{L,D}^{CFD} - C_{L,D}^{EXP}\right|}{C_{L,D}^{EXP}} x100$$
(4)

where e_r is relative error, "EXP" shows the experimental results and "CFD" refers to the simulation results.

4.1 Effects of turbulence models

In validation and verification studies for an HKT; LES, DES, $k-\omega$ SST turbulence models are generally used. In the current study, DES, RANS $k-\omega$ SST and Realizable $k-\varepsilon$ models were used and comparisons were made with the experimental results. While the DES turbulence model is a hybrid model [40] solving LES and RANS models together, the LES model solves the majority of the incoming flow [41]. The RANS model solves the flow separations on the boundary layer and solid surfaces [42]. This model provides a great advantage to researchers.

When the studies conducted by the researchers are examined, it is seen that the maximum relative error for the C_P and C_T does not exceed 8 % for different TSR values in HKT verification studies [23, 40]. In the study conducted by researchers, DES, and RANS $k-\omega$ SST turbulence model comparisons were made [41]. In another study, the authors compared the $k-\omega$ SST and Transient SST (TSST) models. As a result, it was found that TSST provided better results regarding C_P compared to the $k-\omega$ SST turbulence model [43]. In the current study, this comparison was included in the Realizable $k-\varepsilon$ turbulence model. The results found significantly support the CFD studies conducted.

The effect of turbulence models; RANS (k- ω SST and Realizable k- ε) and DES were investigated using the HAHT model under the same operating conditions and the results were compared with the existing experimental data. As a result, 11 % and 4 % relative errors were found for C_P and C_T , respectively, with the RANS k- ω SST model. In a similar study, the RANS k- ω model was used, and the relative error for C_P was found to be between 3 % and 14 % at different TSR values [44]. For the DES model, relative error of 0.5 % and 3 % were calculated in the comparison of C_P and C_T , and when the RANS Realizable k- ε model was used, the relative error values of 3 % and 2 % were found regarding C_P and C_T , respectively. The comparisons between EFD and CFD data in terms of C_P and C_T using different turbulence models are demonstrated in Figure 8. It can be concluded that while DES model gave closer results to the experiments regarding C_P , better results were obtained with Realizable k- ε in terms of C_T values (Figure 13) in the comparisons.

4.2 Effects of rotation methods

In CFD studies of rotating objects, two common rotation methods are typically employed in turbine analysis [18, 24, 38]. These are the MRF and SM methods. In this study, these two methods were performed in the same flow field and conditions for the proper comparison of the hydrodynamic performance of the HAHT model. Compared to the experiments, it has been concluded that the SM method has a relative error of 3 % for C_P and 2 % for C_T , while the MRF method gives worse results regarding the predictions of hydrodynamic performance parameters (relative error value of 23 % for C_P and 12 % for C_T). In the study, the C_P was found to have a maximum of 30 % relative error at TSR=4.8 and less 17 % in other TSR values than compared to the experimental study [44]. Although the relative error for the thrust can be acceptable for the MRF method, the power coefficient could not be predicted in a reasonable range compared with experiments. In a similar study, the MRF method and RANS model were used to compare CFD with experimental results, and it is seen that the C_P consistently predicts at the peak points, while the C_T is not estimated properly [41]. This is critical, as MRF can only be reliably used near the turbine's optimal operating point. As a result of the current study and case studies, it is seen that the SM method is more consistent in predicting the hydrodynamic performances of HKTs compared to the MRF method. In another study, it is shown that the SM method makes closer predictions to the experimental results than the MRF method [24]. Figure 14 presents the comparisons between EFD and CFD results including the effects of the rotation method for the predictions of C_P and C_T values, respectively.



Fig. 13 The effects of turbulence models regarding predictions of C_P and C_T respectively



Fig. 14 The effects of rotation methods regarding predictions of C_P and C_T , respectively



Fig. 15 The effects of mesh density and y^+ regarding predictions of C_P and C_T , respectively

4.3 Effects of mesh density and y^+

Studies have shown that different mesh types and a total number of cells are used for the CFD calculations [21, 35]. Researchers conducted the analyses at different y^+ values [18, 36]. In this study, the y^+ change was examined by simply increasing the total number of cells (N) in the same turbulence model and

with the same mesh quality. While performing the analyses, the Realizable *k*- ε turbulence model was used as the turbulence model and was compared with the experimental data. As a result of this comparison, the minimum relative error was calculated for C_P as 3 % and C_T as 2 % at the y^+ value 4. As can be seen from the results, the number of cells on the turbine geometry and the y^+ effect are more pronounced for C_P , while it does not change much for C_T . The mesh density and y^+ study were conducted at TSR 3, PA 0°, and 2 m/s velocity inlet conditions, and the results were compared with the experiments [33]. Figure 15 presents the effects of the total number of cells and the y^+ on the hydrodynamic performance of the turbine. As can be understood from Figure 15, it is observed that as the y^+ value approaches 1, it gives more consistent results to the experiments in terms of C_P , while there is not a huge change regarding C_T values.

4.4 Comparison of experimental and CFD results for HAHT

Based on the analysis of how different parameters affect the CFD predictions of the hydrodynamic performances for the HKTs, the y^+ value was determined as 4; the DES model was chosen as the turbulence model, and the SM approach was decided as the rotation method. These choices appear to have the minimum relative error value compared to the experiments. The CFD analyses of the turbine model used in the experimental study were carried out at different TSR values and operating conditions with a PA of 0°, 4°, and 8°. The comparative CFD computations were conducted with the finest mesh density with a total number of cells of 12.6 M.



Fig. 16 Comparison between EFD and CFD results regarding C_P

When the experimental study on the HKT with various pitch angles (0°, 4°, and 8°) at the same TSR values is examined, it is seen that the HKT with a 4° of pitch angle provides the highest C_P . It means that the C_P does not always increase as the pitch angle increases. The C_P has its most extreme point at 4°, and it is observed that the less C_P value is at a higher pitch angle. This change makes the pitch angle important for the HKTs design [33]. As a result of the analysis of different TSR values of the turbine with a PA of 0°, the maximum relative error value for the C_P value, the maximum relative error value was found to

be 7 %. Figure 16 shows the comparison of CFD analysis results at different TSR values with experimental results regarding C_P for 0°, and 8° PA, respectively.

As a result of the analysis of different TSR values of the turbine with a PA of 8°, the maximum relative error value for the C_P value was calculated to be 11 % when compared with the experimental values. In the C_T value, the maximum relative error was found to be 14 %.

It can be seen from Figure 16 and 17, the maximum C_P value could be reached between 3 and 4 TSR while the C_T value was increasing with the increment of the TSR value for both PA. The C_P value could be predicted more consistently for 0° PA compared to 8°. While the C_T prediction shows a good agreement for each TSR value for 0°, after 5 TSR value, the gap between CFD and EFD results demonstrates a receding trend for 8° PA.



Fig. 17 Comparison between EFD and CFD results regarding C_T

4.5 Pressure and velocity distributions on the HAHT

Figure 18 presents the pressure distribution on turbine blades for different PA (4° and 8°) at the suction side and perspective view (right). During the tests, the change of the PA was examined, and it was concluded that C_T was significantly reduced by increasing the PA while the C_P was not affected too much by this change (Figure 18). When the PA was 0°, the force on the blade contributed more to the thrust, while for 8°, the increased PA resulted in a reduced AoA and, hence, lower thrust on the turbine (Figure 17).

Figure 19 also presents the velocity distribution around the foil section of the turbine blade with velocity streamlines around and different PA positions. It was concluded that, while the PA increased, velocity fields underwent dramatically changes at the lower part of the blade section. This change also supports reducing the thrust value while the PA was increasing from 0° to 8° (Figure 19; from top to bottom). It can also be seen from this figure, that the velocity distribution changes around the leading and trailing edge of the turbine blades with the change in the PA.



Fig. 18 Pressure distribution on blades @ TSR 4 (top - 4°, bottom - 8°)

To sum up, the above CFD analysis results demonstrated that the prediction of the C_P would be inaccurate for the operating conditions with larger PA (Figure 16 - 8°) due to the large flow separations around the blades. This phenomenon is especially vital for turbine blades designed based on the stall regulation while these HKTs are operating most efficiently before stalling and near stalling.





Fig. 19 Velocity distribution and streamlines with different PA (from top to bottom - 0°, 4° and 8°)

The streamlines around and behind the turbine are shown in Figures 20 and 21. While the PA is 8° , vortex formation occurs behind and around the HKT, while the PA is 4° , vortex formation is not observed behind the HKT. As a result, HKT performs better at PA 4° in terms of C_P . In the geometry of a HKT with a PA of 8° , it is observed that the decreases in the C_T are caused by vortices formed behind and around it.

Brodogradnja Volume 76 Number 3 (2025) 76308



Fig. 21 Velocity distribution and streamlines around the turbine (TSR=4)

4.6 Investigation of turbulence models in terms of isosurface of the Q-criterion

Finally, Figure 22 presents the isosurface of the Q-criterion with Z velocity (aligned with the flow) comparing RANS (k- ω SST) and DES approaches while the DES gives closer results to the experiments regarding C_P and C_T values (Figure 13).



Fig. 22 Comparison between DES and RANS regarding isosurface of Q-criterion with Z velocity (top - RANS k- ω SST, bottom - DES)

5. Conclusion and future works

The inferences from CFD analysis of the HAHT model with a PA of 0° :

- It is seen that the CFD analysis results are very close to the experimental results in cases where it produces maximum torque and maximum C_P at TSR 3 and 4 values. The maximum relative error value was found to be 1 % in the C_P .
- It was concluded that the CFD analysis results regarding C_T are close to the experimental results for each TSR value (the maximum relative error value was found to be 7 %)
- The reason why relative error rates are high at extreme TSR values is thought to be that low C_P values are difficult to predict with CFD analyses.

The inferences from CFD analysis of the HAHT model with a PA of 8°:

- It was shown that C_P values are obtained at a constant relative error rate at every *TSR* value, except for the extreme TSR values. The maximum relative error was found to be 11 % in the C_P value.
- It is seen that at each TSR value, the C_T values are close to the experiments (the maximum relative error was found to be 14 %).
- As a result of the uncertainty analysis performed in the experiments, the deviation rate for the C_P was found to be 2.9 %, and the deviation rate for the C_T was found to be 0.9 % for the PA of 8° [33]. The CFD results can be considered to be reasonable based on these relative error rates

during the experiments.

Evaluating the current study in general, CFD analyses of the foil profiles used in HKT geometries were performed, and as a result, reasonable estimates were made as C_L , while it seems that it is difficult to predict in terms of C_D , especially due to the vortices formed behind the foil profiles at stall angle and high AoA.

Turbulence modelling, rotation method, mesh density, and y^+ studies were performed to predict the hydrodynamic performance of a HKT properly. As a result, the DES method for modelling turbulence, SM method as a rotation method and $y^+=4$ value was determined. After these admissions, analyses were performed on different TSR values and compared with the experimental results. Although consistent results were found in the optimal operating condition (TSR=4) of HKT, very reasonable results could not be found at higher TSR values and pitch angles. It was shown that this is due to vortices that form around the turbine at high pitch angles. In general, consistent results were obtained when comparing the experiments with the CFD results, and the study has achieved its goal.

Building on the evaluation of the hydrodynamic performance of the HAHT model using both EFD and CFD approaches, future studies could focus on developing a more efficient turbine model with the goal of enhancing the power coefficient (CP) through optimization techniques. Additionally, The CFD prediction methods can be developed using adaptive mesh approaches with more complex turbulence models (LES i.e.) for better predictions and comparisons.

ACKNOWLEDGEMENTS

The present investigation was partly based on the results of a PhD thesis entitled "The Design and Optimization of the Hydrokinetic Turbines" developing based at Bursa Technical University, Maritime Faculty, Naval Architecture and Marine Engineering Department.

REFERENCES

[1] Zhang, Y., Li, D., Hong, S., Zhang, M., 2023. Design of a new oscillating-buoy type wave energy converter and numerical study on its hydrodynamic performance. *Brodogradnja*, 74(1), 145-168. <u>https://doi.org/10.21278/brod74108</u>

[2] Alp Erkurtulmus, S., Pesman, E., 2024. GIS-based floating offshore wind turbine installation site selection using fuzzy analytic hierarchy process in northeast Aegean Sea. *Brodogradnja*, 75(2), 1-21. <u>https://doi.org/10.21278/brod75204</u>

[3] Clausen, P., Whale, J., Wood, D., 2021. Small wind and hydrokinetic turbines. *The Institution of Engineering and Technology*. https://doi.org/10.1049/PBPO169E

[4] Talukdar, P. K., 2019. In-situ experiments and numerical simulation of vertical-axis hydrokinetic turbines for small-scale power generation. *Doctoral dissertation*, Indian Institute of Technology Guwahati, Guwahati, Assam, India.

[5] Bhagat, R., Kumar, D., Sarkar, S., 2021. Employability of vertical axis crossflow whirlybird rotor as hydrokinetic turbine and its performance prediction corresponding to different design parameters. *Ocean Engineering*, 238, 109744. https://doi.org/10.1016/j.oceaneng.2021.109744

[6] Saini, G., Saini, R. P., 2019. A review on technology, configurations, and performance of cross-flow hydrokinetic turbines. *International Journal of Energy Research*, 43(13), 6639-6679. <u>https://doi.org/10.1002/er.4625</u>

[7] Sarma, N. K., Biswas, A., Misra, R. D., 2014. Experimental and computational evaluation of Savonius hydrokinetic turbine for low velocity condition with comparison to Savonius wind turbine at the same input power. *Energy Conversion and Management*, 83, 88-98. <u>https://doi.org/10.1016/j.enconman.2014.03.070</u>

[8] Cardoso, R. L. B., Ramos, R. P. B., Filha, E. M. L., Ribeiro, M. M., Candido, V. S., Rodrigues, J. D. S., et al., 2024. Modelling and analysis of jute fiber reinforced epoxy composite in the development of wind blade for low intensity winds. *Journal of Materials Research and Technology*, 28, 3619-3630. <u>https://doi.org/10.1016/j.jmrt.2023.12.151</u>

[9] Chhari, K., Raj, U., Galav, A., Dhillon, L., Tiwari, P., Singh, J. P., 2023. Aerodynamic and bending analysis of low-speed airfoils at high reynold number. *Materials Today: Proceedings*, 72, 1524-1529. <u>https://doi.org/10.1016/j.matpr.2022.09.381</u>

[10] Aboelezz, A., Ghali, H., Elbayomi, G., Madboli, M., 2022. A novel VAWT passive flow control numerical and experimental investigations: Guided Vane Airfoil Wind Turbine. *Ocean Engineering*, 257, 111704. https://doi.org/10.1016/j.oceaneng.2022.111704

[11] Romani, S., Morgut, M., Parussini, L., Piller, M., 2025. Uncertainty quantification and global sensitivity analysis of turbulence model closure coefficients for sheet cavity flow around a hydrofoil. *Brodogradnja*, 76(1), 1-16. <u>https://doi.org/10.21278/brod76105</u>

[12] Salleh, M. B., Kamaruddin, N. M., Mohamed-Kassim, Z., 2022. Experimental investigation on the effects of deflector angles on the power performance of a Savonius turbine for hydrokinetic applications in small rivers. *Energy*, 247, 123432. https://doi.org/10.1016/j.energy.2022.123432

[13] Ramadan, A., Hemida, M., Abdel-Fadeel, W. A., Aissa, W. A., Mohamed, M. H., 2021. Comprehensive experimental and numerical assessment of a drag turbine for river hydrokinetic energy conversion. *Ocean Engineering*, 227, 108587. https://doi.org/10.1016/j.oceaneng.2021.108587

[14] Maldar, N. R., Yee, N. C., Oguz, E., Krishna, S., 2022. Performance investigation of a drag-based hydrokinetic turbine considering the effect of deflector, flow velocity, and blade shape. *Ocean Engineering*, 266, 112765. https://doi.org/10.1016/j.oceaneng.2022.112765

[15] Schleicher, W. C., Riglin, J. D., Oztekin, A., 2015. Numerical characterization of a preliminary portable micro-hydrokinetic turbine rotor design. *Renewable Energy*, *76*, 234-241. <u>https://doi.org/10.1016/j.renene.2014.11.032</u>

[16] Silva, P. A. S. F., Shinomiya, L. D., de Oliveira, T. F., Vaz, J. R. P., Mesquita, A. L. A., Junior, A. C. P. B., 2017. Analysis of cavitation for the optimized design of hydrokinetic turbines using BEM. *Applied energy*, *185*, 1281-1291. https://doi.org/10.1016/j.apenergy.2016.02.098

[17] Niebuhr, C. M., Schmidt, S., Van Dijk, M., Smith, L., Neary, V. S., 2022. A review of commercial numerical modelling approaches for axial hydrokinetic turbine wake analysis in channel flow. *Renewable and Sustainable Energy Reviews*, *158*, 112151. <u>https://doi.org/10.1016/j.rser.2022.112151</u>

[18] Aguilar, J., Velásquez, L., Romero, F., Betancour, J., Rubio-Clemente, A., Chica, E., 2023. Numerical and experimental study of hydrofoil-flap arrangements for hydrokinetic turbine applications. *Journal of King Saud University-Engineering Sciences*, 35(8), 577-588. <u>https://doi.org/10.1016/j.jksues.2021.08.002</u>

[19] Lee, J., Kim, Y., Khosronejad, A., Kang, S., 2020. Experimental study of the wake characteristics of an axial flow hydrokinetic turbine at different tip speed ratios. *Ocean Engineering*, 196, 106777. <u>https://doi.org/10.1016/j.oceaneng.2019.106777</u>

[20] Nago, V. G., dos Santos, I. F. S., Gbedjinou, M. J., Mensah, J. H. R., Tiago Filho, G. L., Camacho, R. G. R., et al., 2022. A literature review on wake dissipation length of hydrokinetic turbines as a guide for turbine array configuration. *Ocean Engineering*, 259, 111863. <u>https://doi.org/10.1016/j.oceaneng.2022.111863</u>

[21] Riglin, J., Carter III, F., Oblas, N., Schleicher, W. C., Daskiran, C., Oztekin, A., 2016. Experimental and numerical characterization of a full-scale portable hydrokinetic turbine prototype for river applications. *Renewable Energy*, 99, 772-783. https://doi.org/10.1016/j.renene.2016.07.065

[22] Abutunis, A., Taylor, G., Fal, M., Chandrashekhara, K., 2020. Experimental evaluation of coaxial horizontal axis hydrokinetic composite turbine system. *Renewable Energy*, 157, 232-245. <u>https://doi.org/10.1016/j.renene.2020.05.010</u>

[23] El Fajri, O., Bowman, J., Bhushan, S., Thompson, D., O'Doherty, T., 2022. Numerical study of the effect of tip-speed ratio on hydrokinetic turbine wake recovery. *Renewable Energy*, 182, 725-750. <u>https://doi.org/10.1016/j.renene.2021.10.030</u>

[24] Dorella, J. J., Volpe, N. J., Storti, B. A., Albanesi, A. E., Zeitler, F. E., 2023. An automatic parallel scheme to design an augmented hydrokinetic river turbine using a simulation-based optimization approach. *Ocean Engineering*, 268, 113374. https://doi.org/10.1016/j.oceaneng.2022.113374

[25] Kale, F. M., Yilmaz, N., Sokmen, K. F., 2024. A Numerical Investigation of Airfoil Sections for More Efficient Horizontal Axis Hydrokinetic Turbine Design. *Global Maritime Congress*, 311-316.

[26] Koyuncuoğlu, A. C., Yilmaz, O., Oğuş, M. Ü., 2018. Numerical investigation of aerodynamic characteristics of low Reynolds number airfoils S826 and S809. *In Proceedings of the 6th International Eurasian Conference on Science, Engineering and Technology*, 22-23 November, Ankara, Turkey, 2276-2284.

[27] Celik, I. B., Ghia, U., Roache, P. J., Freitas, C. J., 2008. Procedure for estimation and reporting of uncertainty due to discretization in CFD applications. *Journal of Fluids Engineering-Transactions of the ASME*, 130(7). https://doi.org/10.1115/1.2960953

[28] Selig, M.S., Guglielmo, J.J., Broeren, A.P., Giguere, F., 1995. Summary of Low-Speed Airfoil Data: Vol.1. Soartech Publications.

[29] Abutunis, A., Hussein, R., Chandrashekhara, K., 2019. A neural network approach to enhance blade element momentum theory performance for horizontal axis hydrokinetic turbine application. *Renewable Energy*, 136, 1281-1293. https://doi.org/10.1016/j.renene.2018.09.105

[30] Wang, D., Atlar, M., Sampson, R., 2007. An experimental investigation on cavitation, noise, and slipstream characteristics of ocean stream turbines. *Proceedings of the Institution of Mechanical Engineers, Part A: Journal of Power and Energy*, 221(2), 219-231. <u>https://doi.org/10.1243/09576509JPE310</u>

[31] Shi, W., Wang, D., Atlar, M., Seo, K. C., 2013. Flow separation impacts on the hydrodynamic performance analysis of a marine current turbine using CFD. *Proceedings of the Institution of Mechanical Engineers, Part A: Journal of Power and Energy*, 227(8), 833-846. <u>https://doi.org/10.1177/0957650913499749</u>

[32] Barbarić, M., Guzović, Z., 2020. Investigation of the possibilities to improve hydrodynamic performances of microhydrokinetic turbines. *Energies*, 13(17), 4560. <u>https://doi.org/10.3390/en13174560</u> [33] Shi, W., Rosli, R., Atlar, M., Norman, R., Wang, D., Yang, W., 2016. Hydrodynamic performance evaluation of a tidal turbine with leading-edge tubercles. *Ocean Engineering*, 117, 246-253. <u>https://doi.org/10.1016/j.oceaneng.2016.03.044</u>

[34] Atlar, M., 2011. Recent upgrading of marine testing facilities at Newcastle University. *In the Proceedings of the 2nd International Conference on Advanced Model Measurement Technology for the Eu Maritime Industry*, 4-6 April, Newcastle, United Kingdom.

[35] Song, S., Demirel, Y. K., Atlar, M., Shi, W., 2020. Prediction of the fouling penalty on the tidal turbine performance and development of its mitigation measures. *Applied Energy*, 276, 115498. <u>https://doi.org/10.1016/j.apenergy.2020.115498</u>

[36] Tampier, G., Troncoso, C., Zilic, F., 2017. Numerical analysis of a diffuser-augmented hydrokinetic turbine. *Ocean engineering*, 145, 138-147. <u>https://doi.org/10.1016/j.oceaneng.2017.09.004</u>

[37] Ansys Theory Guide, 2021.

https://dl.cfdexperts.net/cfd_resources/Ansys_Documentation/Fluent/Ansys_Fluent_Theory_Guide.pdf. accessed 11st February 2025

[38] Barbarić, M., Batistić, I., Guzović, Z., 2022. Numerical study of the flow field around hydrokinetic turbines with winglets on the blades. *Renewable Energy*, 192, 692-704. <u>https://doi.org/10.1016/j.renene.2022.04.157</u>

[39] Posa, A., Broglia, R., 2021. Characterization of the turbulent wake of an axial-flow hydrokinetic turbine via large-eddy simulation. *Computers & Fluids*, 216, 104815. <u>https://doi.org/10.1016/j.compfluid.2020.104815</u>

[40] Wang, W. Q., Song, K., Yan, Y., 2019. Influence of interaction between the diffuser and rotor on energy harvesting performance of a micro-diffuser-augmented hydrokinetic turbine. *Ocean Engineering*, *189*, 106293. https://doi.org/10.1016/j.oceaneng.2019.106293

[41] Abutunis, A., Fal, M., Fashanu, O., Chandrashekhara, K., Duan, L. (2021). Coaxial horizontal axis hydrokinetic turbine system: Numerical modeling and performance optimization. *Journal of Renewable and Sustainable Energy*, *13*(2), 024502. <u>https://doi.org/10.1063/5.0025492</u>

[42] Menter, F. R., 1994. Two-equation eddy-viscosity turbulence models for engineering applications. *AIAA journal*, *32*(8), 1598-1605. <u>https://doi.org/10.2514/3.12149</u>

[43] Benavides-Morán, A., Rodríguez-Jaime, L., Laín, S., 2021. Numerical investigation of the performance, hydrodynamics, and free-surface effects in unsteady flow of a horizontal axis hydrokinetic turbine. *Processes*, 10(1), 69. https://doi.org/10.3390/pr10010069

[44] Ferraiuolo, R., Pugliese, F., Álvarez, E. Á., Yosry, A. G., Giugni, M., Del Giudice, G., 2024. Experimental and numerical investigation of a three-blade horizontal axis hydrokinetic water turbine (HAHWT) in high blockage conditions. *Renewable Energy*, 237, 121640. <u>https://doi.org/10.1016/j.renene.2024.121640</u>