



UNIONE EUROPEA
Fondo Sociale Europeo



UNIVERSITÀ
DEGLI STUDI
FIRENZE
Da un secolo, oltre.

DOTTORATO DI RICERCA IN Ingegneria Industriale

CICLO XXXVII - PON ex DM1061

Investigation and Multi-fidelity Modelling of Darrieus Hydrokinetic Turbines Indagine e Modellizzazione Multi-Fidelity di Turbine Idrocinetiche Darrieus

Settore Scientifico Disciplinare
ING-IND/09

Dottorando

Dott. Mohamed Omar Sherif

Supervisore

Prof. Bianchini Alessandro

Coordinatore

Prof. Giovanni Ferrara

Salvo eventuali più ampie autorizzazioni dell'autore, la tesi può essere liberamente consultata e può essere effettuato il salvataggio e la stampa di una copia per fini strettamente personali di studio, di ricerca e di insegnamento, con espresso divieto di qualunque utilizzo direttamente o indirettamente commerciale. Ogni altro diritto sul materiale è riservato.

“The important thing is not to stop questioning”

Albert Einstein

A Rola

DECLARATION

I hereby declare that this submission is my own work and, to the best of my knowledge and belief, it contains no material previously published or written by another person, nor material which to a substantial extent has been accepted for the award of any other degree or diploma at University of Florence or any other educational institution, except where due references are provided in the thesis itself.

Any contribution made to the research by others, with whom I have been working at the University of Florence or elsewhere, is explicitly acknowledged in the thesis.

Omar Sherif Mohamed

17 March 2025

ACKNOWLEDGEMENTS

I would like to express my heartfelt gratitude to all those who have supported me throughout this challenging yet rewarding doctoral journey.

First and foremost, I am deeply indebted to Professor Giovanni Ferrara for his unwavering guidance and encouragement from the very inception of this research.

My sincere appreciation goes to Professor Alessandro Bianchini for warmly welcoming me into the research group and providing invaluable assistance as I navigated a new academic and cultural landscape. His insightful perspectives and constant support have been instrumental in shaping this work.

I am especially grateful to Dr Francesco Balduzzi for his patience, enriching technical discussions, and continuous encouragement. His support was crucial in helping me reach this milestone.

I extend my profound thanks to Dr Pier Francesco Melani, whose mentorship from the very beginning was fundamental to my research endeavours. His thoughtful suggestions, insightful reading recommendations, and technical guidance played a pivotal role in my doctoral journey.

I am also thankful to my colleagues in the REASE group, who have been not only supportive collaborators but also dear friends, fostering a warm and welcoming environment during my time away from home.

Lastly, my deepest gratitude goes to my family and friends for their unwavering love, encouragement, and sacrifices. Their belief in me has been the foundation of all my accomplishments.

ABSTRACT

Hydrokinetic energy has gained attention recently due to its low environmental impact and cost-effectiveness. In particular, Darrieus-type Hydrokinetic Turbines (HKTs) are well suited for rivers and small watercourses because of their compact design, simple blade structure, and ease of operation, with no need for pitching or yawing adjustments. These turbines also demonstrate efficiency improvements when arranged in closely spaced arrays. However, accurately modelling their performance remains challenging due to complex hydrodynamics, unsteady flow behaviour, and interactions with the surrounding environment, including channel walls and the free surface. While analytical models fail to capture these complexities, high-fidelity simulations are computationally prohibitive. This thesis addresses these gaps by advancing the understanding of Darrieus turbine interactions within arrays and with hydraulic environments while enhancing modelling capabilities for real-world scenarios. The thesis is based on a collection of six peer-reviewed journal articles addressing these challenges. Numerical models with varying fidelity were developed using Computational Fluid Dynamics (CFD) in Ansys Fluent®. Initially, two-dimensional blade-resolved unsteady RANS simulations were employed to investigate mechanisms driving efficiency gains in turbine arrays. Results revealed that mutual interaction zones suppress inflow deviation, increasing blade angles of attack and enhancing performance. The Actuator Line Method (ALM) was then tuned as a medium-fidelity tool for modelling Darrieus rotors without resolving blade boundary layers. ALM accurately captured twin-rotor interactions at medium and high tip-speed ratios (TSRs), predicting blade loads, torque, and wake structures. A comprehensive three-dimensional model was subsequently developed by coupling ALM with the Volume of Fluid (VOF) method to simulate free surface effects. Additionally, a more computationally efficient model using the Actuator Cylinder (AC) concept was created for quasi-steady-state analysis. To validate the computational model, an experimental campaign was conducted by the industrial partner, HE-PowerGreen. Field measurements were obtained for the flow field characteristics and the performance of an array composed of four closely spaced turbines. Both the CFD-ALM-VOF and CFD-AC-VOF models were validated against the experimental data, showing good agreement in flow velocity profiles, turbine performance, and free surface variations. The ALM shows accurate matching with the experimental measurements in terms of both flow field and array performance. On the other hand, the AC model provides reasonably accurate turbine performance predictions and captures array impacts on the channel and flow field, though it overestimates blockage effects and underestimates turbine-to-turbine interactions compared to the more precise ALM. Despite this, the AC model achieves a favourable trade-off between accuracy and computational efficiency, reducing computational time by an order of magnitude. Both models emphasised the critical role of two-way interactions between turbines and the hydraulic environment, including variations in water level and inflow velocity. The findings of this thesis advance the understanding of Darrieus turbine behaviour in arrays and complex hydraulic conditions. The developed models enhance predictive capabilities, offering valuable tools for academia and industry to optimise hydrokinetic energy systems.

Keywords: Darrieus, Computational Fluid Dynamics, Hydrokinetic Energy, Actuator Cylinder, Actuator Line Method

PROPOSITIONS

Motivation

- Hydrokinetic energy presents a promising low-cost renewable energy technology with minimal environmental impact.
- Darrieus turbines offer several advantages. However, lack of understanding of their complex hydrodynamics, turbine-to-turbine interactions within arrays, and array interactions with the channel hinders their widespread implementation.

Original contribution

- Investigated the mechanisms of power augmentation in closely spaced Darrieus turbine arrays, identifying the role of flow interactions between neighbouring turbines.
- Tuned and validated the Actuator Line Method (ALM) model for accurately simulating Darrieus turbines in arrays.
- Developed a novel three-dimensional CFD model coupling the tuned ALM with the Volume of Fluid (VOF) method to simulate the two-way interactions between Darrieus turbines and the channel.
- Developed a computationally efficient quasi-steady-state model using the Actuator Cylinder (AC) concept coupled with VOF, suitable for industrial applications.

Open questions

- How accurate can the Actuator Line Method (ALM) be if an appropriate model for unsteady forces is implemented?
- does the two-way channel interactions impact how HKT turbines and their arrays should be designed?
- How crucial is it to adopt CFD-based methods for the design of HKT turbine arrays?

INDEX

1	Introduction.....	1
1.1	Hydrokinetic Energy.....	1
1.2	Darrieus Turbines	2
2	State of the art.....	6
2.1	Darrieus turbine in arrays.....	6
2.1.1	Experimental Investigations	6
2.1.2	Numerical Investigations	8
2.1.3	Discussion	9
2.2	Modelling of Darrieus Turbines	12
2.2.1	Double-Multiple Streamtube (DMST).....	13
2.2.2	Actuator Cylinder (AC)	14
2.2.3	Lifting Line Theory (LLT).....	15
2.2.4	Actuator Line Method (ALM).....	16
2.2.5	Computational Fluid Dynamics (CFD)	17
2.2.6	Discussion	18
2.3	Darrieus turbines in hydrokinetic energy	21
2.3.1	HKT-Channel Two-way Interactions	21
2.3.2	Volume of Fluid (VOF).....	23
2.3.3	Discussion	25
3	Original contribution.....	27
3.1	List of Publications.....	27
3.2	Discussion	28
3.2.1	Physics of Darrieus turbine in arrays	28
3.2.2	ALM tuning in open-field conditions	32
3.2.3	Modelling and validation of Darrieus HKTs under realistic conditions.....	36
	Conclusions.....	145

LIST OF FIGURES

Figure 1.1: Different types of vertical axis hydrokinetic turbines (adopted from [12])	2
Figure 1.2: Schematic illustration for the flow around Darrieus rotor.	3
Figure 2.1: Illustration of terminologies used in clustering the turbines: (a) counter-rotating and co-rotating schemes, (b) modes of interaction: leeward and windward, (c) different layout types: in-line and staggered.....	7
Figure 2.2: Performance mapping for the influence of cluster layout in different configurations of arrays of two turbines (adopted from [23])......	8
Figure 2.3: Illustration of the Double-multiple Streamtube (DMST) model (adopted from [36]).	13
Figure 2.4: Actuator cylinder representation of VAT rotors.	14
Figure 2.5: Illustration of the Lifting Line Theory.....	15
Figure 2.6: Illustration of the Actuator Line Method.	16
Figure 2.7: Example of the mesh details required for a proper blade-resolved CFD for a Darrieus turbine, (a) full domain, (b) rotor, (c) aerofoil, (d) leading edge, and (e) trailing edge. Adopted from [63]	17
Figure 2.8: Overview on the numerical models used to simulate the performance of Darrieus turbines along with their fidelity level in modelling rotor performance and surrounding flow field.	19
Figure 2.9: Schematic representations of (a) sloped bed channel without turbine, (b) with turbine, and (c) horizontal bed channel with a turbine.	22
Figure 2.10: Free surface variation simulated using VOF compared to corresponding experimental results (adopted from [82])	24
Figure 3.1: Mean power coefficient obtained for each turbine for different configurations of turbine arrays at TSR = 3 compared to that of the stand-alone turbine.....	29
Figure 3.2: Instantaneous angle of attack α and torque coefficient C_T for different turbines within different arrays.	30
Figure 3.3: Velocity contours and streamlines for the stand-alone turbine cases (a) compared with cases (b) and (e) for arrays of two and three turbines, respectively.	31
Figure 3.4: dimensionless velocity contours showing the Frozen ALM concept used to study different Regularization kernel shapes.	32
Figure 3.5: Bubble chart showing ALM accuracy dependencies.....	33

Figure 3.6: Comparison between the estimations of ALM vs blade-resolved CFD for the load in terms of moment coefficient for the HKT pair at TSR = 3 (left) and the VAWT pair at TSR 2.7 (right).	34
Figure 3.7: Results of the POD wake assessment: BL-CFD vs ALM vs Frozen ALM	35
Figure 3.8: Schematic representation of the differences and similarities between the ALM and AC turbine models used in [P5] and [P6], respectively.	36
Figure 3.9: (a) Tested array configuration for the installed turbines and (b) schematic drawing for the Biffi canal section and the installed Darrieus turbine array.	37
Figure 3.10: Comparison of the normalised power coefficient values between the ALM and AC models for the stand-alone turbine and turbine T2 in the array, against experimental measurements for turbine T2.	38
Figure 3.11: Comparison of longitudinal velocity profiles from the AC and ALM models with experimental data upstream and downstream of the turbine array at TSR = 2.75.	39
Figure 3.12: Dimensionless velocity contours for a stand-alone turbine and a 4-turbine array comparing the Actuator Line Method to the Actuator Cylinder at a Tip Speed Ratio of 2.75.	40

LIST OF TABLES

Table 2.1: Summery of the studies on deploying Darrieus turbines in close proximity.	10
Table 2.2: Summery of the comparative studies assessing different modelling methods for VATs.	19
Table 3.1: Comparison in terms of computational cost between AC and ALM.	41

NOMENCLATURE

Acronyms

AC	Actuator Cylinder
ALM	Actuator Line Method
BEM	Blade Element Momentum
CFD	Computational Fluid Dynamics
DMST	Double-Multiple Streamtube
EU	European Union
HAT	Horizontal Axis Turbine
HAWT	Horizontal Axis Wind Turbine
HKT	Hydrokinetic Turbines
LES	Large Eddy Simulations
LLT	Lifting Line Theory
TSR	Tip-speed ratio
uRANS	Unsteady Reynolds-averaged Navier-Stokes
UDF	User Defined Function
VAT	Vertical Axis Turbine
VAWT	Vertical Axis Wind Turbine
VOF	Volume of Fluid

Greek symbols

α	angle of attack	[deg.]
Γ	Circulations	[m ² s ⁻¹]
λ	Tip-speed ratio	[-]
μ	Dynamic viscosity	[Kg m ⁻¹ s ⁻¹]
ρ	Density	[Kg m ⁻³]
ϑ	Azimuthal Position	[deg.]
ω	Angular velocity	[m s ⁻¹]

Latin symbols

A	Rotor swept area	[m ²]
c	blade chord length	[m]
C _p	Power Coefficient	[-]
P	Power	[Watt]
R	Turbine Radius	[m]
Re _c	Chord-based Reynold's number	[-]
V _∞	Free stream absolute velocity	[m s ⁻¹]
V _{rel}	Relative Velocity	[m s ⁻¹]

SYNOPSIS

Interest in hydrokinetic energy as a non-conventional renewable source has grown significantly. Unlike traditional hydropower technologies, Hydrokinetic Turbines (HKTs) require minimal civil infrastructure, making them cost-effective with low environmental impact. Darrieus turbines are particularly well-suited for applications in rivers and small watercourses, where flow is unidirectional, predictable, and sometimes controllable. These turbines offer unique advantages: their compact design and simple blade structure make them easier to manufacture than horizontal-axis turbines. They are also easier to control during operation, as they do not require pitching or yawing adjustments. Additionally, the generator level is adjustable, allowing it to be positioned in dry areas above the water. Most importantly, Darrieus turbines benefit from a notable efficiency boost when deployed in close proximity to one another.

Despite their advantages, the hydrodynamics of Darrieus turbines are inherently complex and challenging to model. During operation, the blade experiences continuously varying angles of attack and relative velocities, making performance highly unsteady and prone to dynamic stall effects. Moreover, the complications of this machine extend to the array setups, where there is a lack of quantitative analysis on how the performance of a Darrieus turbine improves when clustered in a closely spaced array, with most research in this area focuses on improving overall performance without addressing the mechanisms behind these efficiency gains. Beyond the interactions between turbines within an array, it is crucial in hydrokinetic applications to account for the two-way interactions between the turbine array and the channel itself, including the channel walls and the free water surface. Thus, the level of complexity associated with modelling Darrieus HKT arrays is highly challenging and represents a major issue limiting industrial explorations. While achieving the level of simulation fidelity needed to capture all the associated complexities is computationally prohibitive, analytical models are inadequate for accurately predicting turbine performance across different operating conditions.

To this end, this thesis aims to bridge key knowledge gaps through two main goals: first, to advance the understanding of Darrieus turbine interactions within arrays and with surrounding hydraulic environments. Second, to enhance the modelling capabilities for accurately predicting Darrieus HKT performance under conditions representative of real-world environments. To achieve these goals, various numerical models were developed at different fidelity levels, all based on Computational Fluid Dynamics (CFD) using Ansys Fluent®.

Initially, an extensive investigation was conducted using two-dimensional blade-resolved unsteady Reynolds-Averaged Navier-Stokes (uRANS) simulations to examine the physics of closely spaced Darrieus turbines, focusing on instantaneous variations in local blade kinematics. The objective was to identify the mechanisms responsible for the power augmentation observed in turbine arrays. This analysis began with various configurations of a turbine pair and was subsequently generalised to arrays with multiple turbines. Despite its crucial role in terms of accuracy, conducting blade-resolved uRANS simulations in a three-dimensional domain for an array of turbines within an open channel is computationally prohibitive. To address this, a comprehensive tuning was applied to the Actuator Line Method (ALM), a medium-fidelity tool

developed for simulating Darrieus rotors without explicitly resolving the blade boundary layer, by assessing key factors affecting ALM accuracy. Following this, an in-depth analysis evaluated the ability of the ALM to replicate the effects observed in closely spaced turbines. Building on these foundations, a detailed, accurate, and computationally feasible three-dimensional model was created by coupling the tuned ALM tool with the Volume of Fluid (VOF) method to simulate the free water surface. Finally, a more computationally efficient model was developed using the Actuator Cylinder (AC) concept, which, combined with the VOF method, produced a quasi-steady-state model suitable for industrial applications.

Upon investigating the kinematic data of closely spaced Darrieus turbines, the results revealed that interactions between adjacent turbines create a mutual interaction zone. In this zone, the inflow deviation, typically known as “streamtube” expansion for a stand-alone turbine, is suppressed. This suppression leads to an increase in the local angle of attack on the blades, which, at higher turbine rotational velocities, results in increased lift and enhanced turbine efficiency. This analysis was applied to each rotor in both two- and three-turbine arrays, considering all possible rotational configurations. It was observed that the pattern of power augmentation remains consistent regardless of the number of rotors and is primarily influenced by the location of the mutual interactions along the azimuthal path of the turbines.

The results of the ALM tuning showed that, in the absence of unsteady effects, the ALM can generate a highly accurate wake flow field, nearly identical to that of blade-resolved CFD, provided that the aerodynamic forces are correctly modelled. Additionally, the findings emphasised that improving the dynamic stall model is a key factor in enhancing the accuracy of the ALM. Furthermore, the results demonstrate that the ALM effectively simulates twin-rotor Darrieus turbines, particularly at medium and high tip-speed ratios. It provides reliable predictions of instantaneous blade loads and successfully captures the torque increase resulting from the mutual interaction between adjacent rotors. The ALM also accurately reproduces the flow field around the twin rotors, as illustrated by a Proper Orthogonal Decomposition (POD) analysis of the wake flow field.

Finally, both of the developed comprehensive models, CFD-ALM-VOF and CFD-AC-VOF, were validated against experimental measurements, showing good agreement in terms of flow field velocity profiles, turbine performance, and variations in the free water surface level near the turbine array. A code-to-code comparison between the two models revealed that the model based on the ALM is more reliable at lower tip-speed ratios, where unsteady effects are relevant, while the AC model accurately predicts the mean performance of the turbines at high and medium TSRs. Comparison of the flow fields from both models showed almost identical results, including variations in the free water surface. In general, the results emphasise the importance of two-way interactions between the turbines and the channel, highlighting how variations in water level and inflow velocity affect the efficiency evaluation of hydrokinetic turbines.

The findings of this thesis have contributed to deepening the understanding of how Darrieus turbines interact both within arrays and with the surrounding hydraulic conditions. Additionally, the two developed models, representing different fidelity levels, are considered an improvement in modelling techniques for Darrieus hydrokinetic turbines in scenarios that closely mirror real-world conditions. These findings could be used by both academic and industrial communities to further advance hydrokinetic energy technology in rivers and small watercourses.

This thesis is structured as follows:

1. **Introduction:** Provides an overview of hydrokinetic energy and introduces the specific focus on Darrieus turbines.

2. **State of the art:** Presents a comprehensive review of existing research on power augmentation in Darrieus turbine arrays, modelling methodologies for Darrieus turbines at various fidelity levels, and the application of Darrieus turbines in hydrokinetic energy systems.
3. **Original Contribution:** Outlines the publications resulting from this research and discusses how these publications contribute to the overarching objectives and goals of the thesis.
4. **Conclusion:** Summarises the key findings, their implications, and outlines recommendations for future research direction.

1 INTRODUCTION

1.1 Hydrokinetic Energy

Climate change has emerged as one of the most significant threats to life on Earth, driving a global push toward transitioning from fossil fuels to renewable energy resources. This shift, accelerated by the goals set in the 2015 Paris Agreement, has prompted many nations to explore sustainable alternatives, including the extraction of energy from water streams, commonly known as hydroelectric energy. Hydroelectric power can be harnessed through two main approaches: hydrostatic and hydrokinetic. The hydrostatic method exploits the potential energy of water by storing it in a reservoir to create a pressure head. In contrast, the hydrokinetic method captures the kinetic energy of flowing water, with minimal head loss, similar to how wind energy is harnessed [1]. Hydrokinetic turbines can be deployed in various water environments, including tidal zones, ocean currents, waves, rivers, and even small watercourses [2].

In general, hydrostatic energy is among the most efficient and reliable renewable energy resources. In 2022, the European Union (EU) generated nearly 30% of the total renewable energy generation from hydrostatic resources, and 12% of its overall electricity production [3]. However, environmental and social impacts, along with limited exploitable resources, have confined its use to developed sites, making further expansion unlikely. In contrast, hydrokinetic energy requires no large civil structures and has minimal environmental impact [4], though its power output is currently lower than hydrostatic turbines. Global interest in hydrokinetic turbines is rising, particularly in countries with existing hydrostatic projects or in developing countries with limited energy resources. Despite growing interest, detailed data on global deployments remain scarce [1], though some case studies provide insight into the current state of hydrokinetic energy generation.

For instance, research is emerging on integrating hydrokinetic systems into existing hydrostatic plants in the Amazon and its tributaries, which constitute the largest watercourse by discharge volume. In Brazil, a study explored installing hydrokinetic turbines downstream of a dam [5], while similar research in Colombia demonstrated promising results [6]. This approach is not confined to the Amazon. For example, a study in India proposed deploying 904 hydrokinetic turbines at an existing facility, with a potential output of 26.4 MW [7]. In Lithuania, where dam construction is restricted due to environmental concerns, hydrokinetic turbines have been suggested as an alternative [8]. Similarly, in Ghana and South Africa, hydrokinetic energy is being

explored as a solution for off-grid and rural communities [9,10]. However, not all regions offer significant potential. In Italy, an analysis of 9,376 small watercourses in the Piemonte region found that only 0.7% were suitable for hydrokinetic power generation, with an estimated installed capacity of 8.3 MW [11].

1.2 Darrieus Turbines

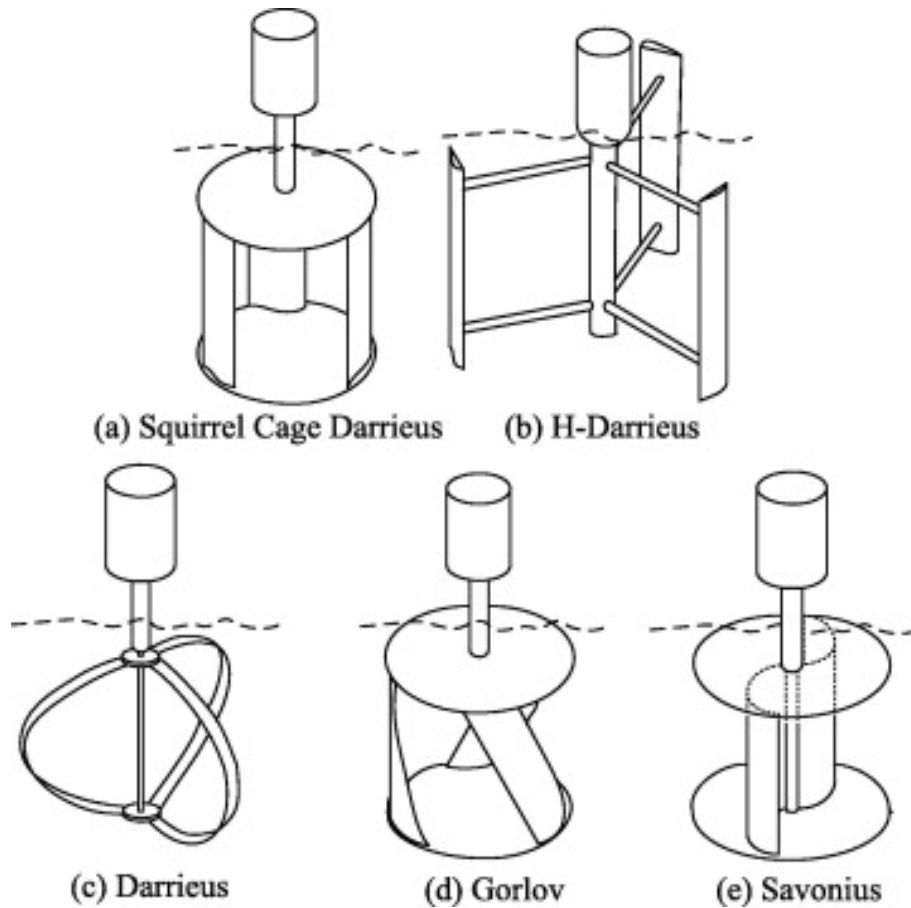


Figure 1.1: Different types of vertical axis hydrokinetic turbines (adopted from [12])

Hydrokinetic energy systems come in various designs and architectures, each suited to specific applications. Similar to wind turbines, hydrokinetic turbines (HKTs) are categorised based on the axis of rotation relative to the flow: horizontal-axis turbines (HATs) and vertical-axis turbines (VATs), which may also be referred to in the literature as “crossflow turbines”. VATs are further divided into lift-driven machines, known as Darrieus turbines, and drag-driven machines, referred to as Savonius turbines. Figure 1.1 illustrates different types of VATs.

While the selection of turbine architecture based on specific applications and design variables is not extensively covered in the literature, HATs are typically employed in large-scale systems, particularly for tidal energy. Conversely, VATs are gaining industrial interest for smaller-scale projects. For applications in small watercourses, VATs offer a promising and reliable solution compared to HATs due to their simpler control mechanisms and the flexibility of positioning powertrains in dry spaces [13]. In contrast, HATs require more complex control systems, with their generators and powertrains needing to be submerged. In addition, recent studies have

demonstrated that clustering Darrieus turbines can enhance overall efficiency compared to individual stand-alone turbines [14,15]. However, while clustering addresses efficiency concerns, it compromises the omnidirectional advantage of the turbines, as the cluster becomes sensitive to changes in inflow direction [16]. Fortunately, this limitation is not a concern in hydrokinetic applications in rivers and small watercourses, where the inflow direction remains constant.

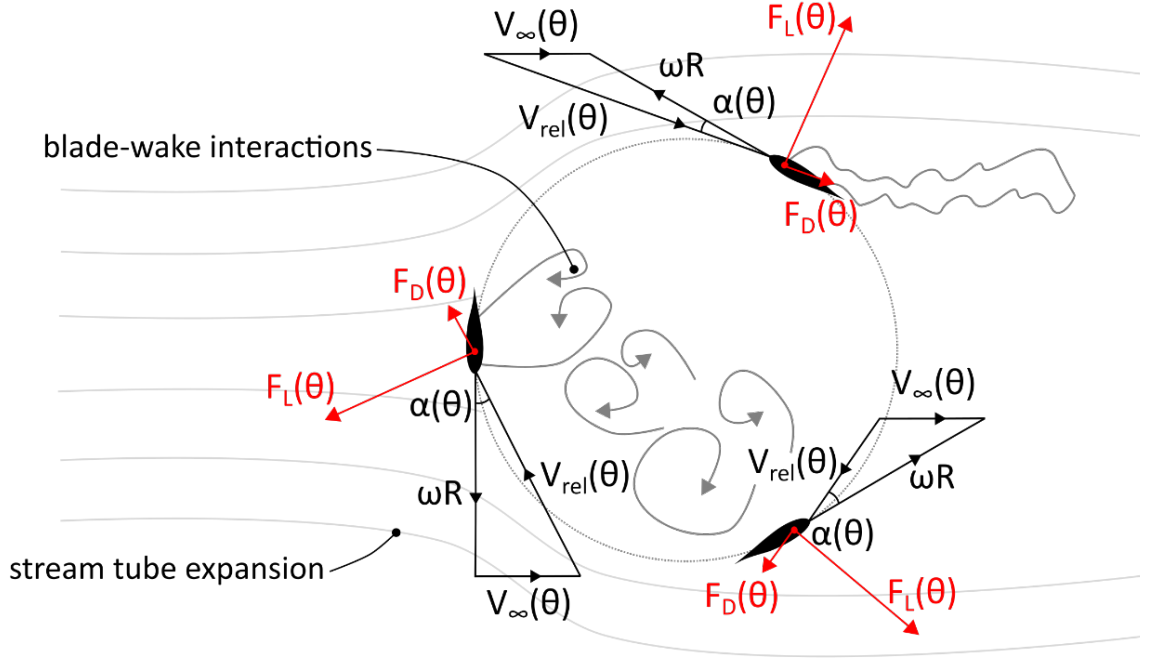


Figure 1.2: Schematic illustration for the flow around Darrieus rotor.

Despite their advantages, the performance of Darrieus turbines is inherently complex due to the instantaneous variation in the blade local kinematics. As shown in Figure 1.2, the relative velocity, V_{rel} and angle of attack, α changes instantaneously as the blade moves along its azimuth path, θ . This variation leads to an unsteady operational regime, which makes the modelling of the turbine forces challenging. In more details, as the tip-speed ratio (TSR), λ , increases, the variation in the angle of attack becomes narrower. TSR is calculated using Eq. 1.1, where ω is the rotor speed in radians per second, R is the rotor radius, and V_∞ is the inflow velocity. This relationship plays a crucial role in optimizing turbine performance, as higher TSR values reduce the dynamic fluctuations in blade forces.

$$\lambda = \frac{\omega R}{V_\infty} \quad 1.1$$

Through a basic kinematic analysis, and assuming a perfectly horizontal absolute flow, α and V_{rel} can be computed as:

$$\alpha = \tan^{-1} \frac{\sin \theta}{\lambda + \cos \theta} \quad 1.2$$

$$V_{rel} = V_{\infty} \sqrt{1 + 2\lambda \cos \theta + \lambda^2} \quad 1.3$$

However, it is important to note that the use of Eqs. 1.2 and 1.3 assumes the absence of what is known as “streamtube expansion”, and it also ignores the influence of blade-wake interactions on the local kinematics. As illustrated in Figure 1.2, streamtube expansion is when the flow onset is subjected to blockage effect due to the presence of the turbine rotor, causing a velocity deficit and a deviation of the inflow. The blade-wake interaction on the other hand is a phenomenon at which the wake of a blade is interacting with the other blades, at which its influence is governed by various factors, including the load unsteadiness in the first place.

One of the important parameters in analysing the aero/hydrodynamic performance is the chord-based Reynolds number Re_c as described in Eq. 1.4:

$$Re_c = \frac{\rho V_{rel} c}{\mu} \quad 1.4$$

Where c is turbine chord length in m, ρ and μ are the flow density and dynamic viscosity in kg/m^3 and kg/ms , respectively. The value of the Re_c usually plays an important role in determining the forces acting on the turbine blade. For VATs, especially those for small scale applications, the range of Re_c variation might be within small values e.g., $10^4 - 10^5$, resulting in different aero/hydrodynamic phenomenon compared to high Reynolds number operation.

To describe the turbine performance, a dimensionless parameter known as power coefficient C_p represents the power efficiency of the turbine and can be computed as Eq. 1.5.

$$C_p = \frac{P}{0.5 \rho A V_{\infty}^3} \quad 1.5$$

Where P is the turbine power in watt and A is the turbine swept area in m^2 . Usually, for a complete turbine performance description, the power coefficient is plotted against the tip-speed ratio.

The latter performance parameters are similar for both wind and hydrokinetic turbines. However, in hydrokinetic applications, the extractable power is limited by interactions between the turbine and the water resource. The turbines impose a blockage effect that influences the hydraulics of the channel, which, in turn, alters the actual performance of the turbines compared to calculations that do not account for turbine-to-channel interactions. This aspect adds an additional layer of complexity, making it more challenging to model HKTs under realistic conditions, where low-fidelity, so-called engineering models do not provide reliable results and could lead to design miscalculations.

2 STATE OF THE ART

This chapter reviews the state-of-the-art of Darrieus-type turbines for hydrokinetic applications; in particular, the performance of Darrieus turbine in grouped clusters, its modelling techniques with emphasis on hydrokinetic applications. It begins by highlighting the key studies on clustering Darrieus turbines in close proximity, drawing insights from wind and hydrokinetic research to discuss optimisation strategies and efficiency gains from such setup. The chapter also explores modelling approaches for Darrieus turbines, ranging from detailed Computational Fluid Dynamics (CFD) to faster, simplified models, and their role in advancing the field. Lastly, it delves into literature on modelling Darrieus turbines for hydrokinetic applications, focusing on integrating models to address blockage effects and free water surface interactions.

2.1 Darrieus turbine in arrays

This section reviews studies on Darrieus turbines deployed in close proximity to form an array, covering both experimental and numerical investigations. It focuses on layout optimization, the physical mechanisms behind performance enhancement compared to stand-alone deployment, key factors influencing performance, and the impact on wake characteristics. A discussion follows, highlighting the significance of the findings in the literature, their current and potential applications, and identifying research gaps, along with an explanation of how this thesis addresses those gaps.

2.1.1 Experimental Investigations

Several studies have demonstrated that the aero/hydrodynamic efficiency of Darrieus turbines can be enhanced if two or more turbines are deployed in close proximity, with respect to a stand-alone turbine. In 2011, Dabiri [14] was the first to report this observation through a field experiment that involved six full-scale Vertical Axis Wind Turbines (VAWTs) arranged in various configurations to investigate counter-rotating, closely spaced turbines. He concluded that the overall efficiency of the turbines could increase by up to 10% if the layout is well-optimised, particularly when the turbines are placed in-line. Subsequently, another study by the same group [17], using field measurements of 18 counter-rotating turbines of the same type, focused on examining the wake characteristics of this compact arrangement. The study concluded that the wake recovers to 95% of the mainstream flow after a distance equivalent to six rotor diameters, which is almost half the recovery distance required for a typical Horizontal Axis Wind Turbine (HAWT).

The latter two studies paved the way for numerous investigations into optimizing the layout of closely deployed Darrieus turbines for improved efficiency and wake recovery. Possible configurations for a turbine pair are illustrated in Figure 2.1-a, where the turbines can either co-rotate in the same direction or counter-rotate. The counter-rotating setup can be classified into two distinct configurations, depending on the mode of mutual interaction between the adjacent turbines, which can be either on the leeward or the windward sides, as illustrated in Figure 2.1-b. Moreover, two types of layouts could be used to cluster Darrieus turbines as illustrated in Figure 2.1-c; the in-line layout, at which all the turbines are on the same lateral axis, while the staggered layout is when the turbines are placed on different lateral axis with a certain oblique angle.

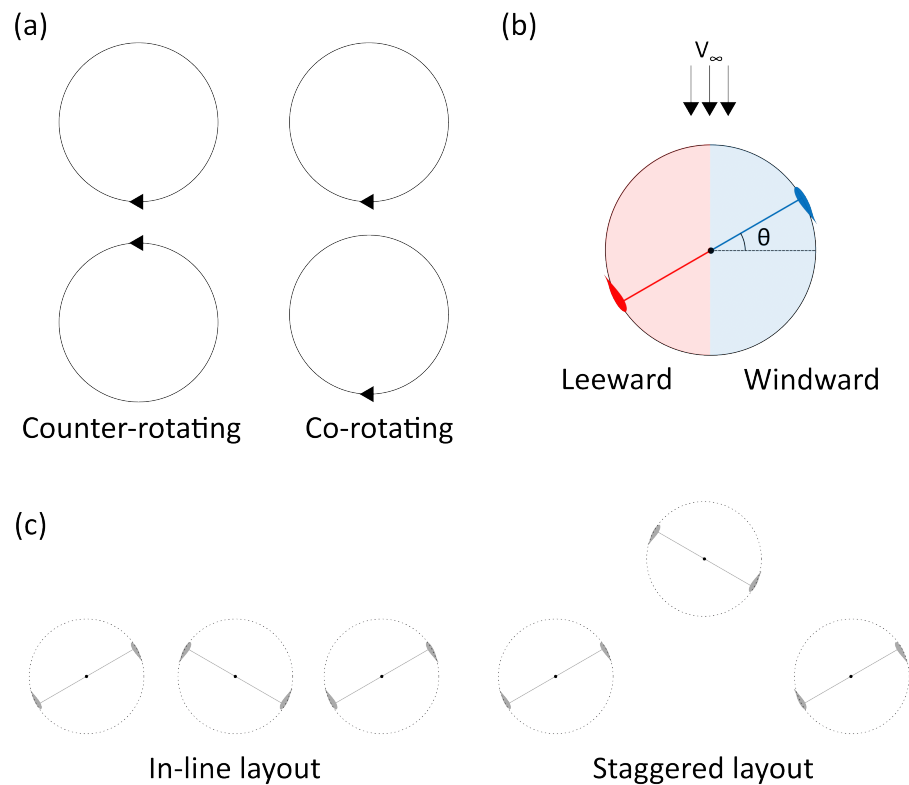


Figure 2.1: Illustration of terminologies used in clustering the turbines: (a) counter-rotating and co-rotating schemes, (b) modes of interaction: leeward and windward, (c) different layout types: in-line and staggered.

Vergaerde et al. [18] conducted wind tunnel tests on a pair of VAWTs, identifying an optimal centre-to-centre spacing of 1.2–1.3 rotor diameters to achieve maximum efficiency, with a 16% increase compared to a stand-alone turbine. Li et al. [19] investigated twin Darrieus turbines in an urban environment, examining the effects of rotor solidity ($\sigma = 0.29, 0.441$, and 0.588) and spacing ($1.5D$ to $3D$). They found that higher rotor solidity and closer spacing improved efficiency, although higher solidity may reduce stand-alone turbine performance, making lower solidity setups potentially more efficient overall in absolute terms.

Müller et al. [20] experimentally investigated wake dynamics of paired Darrieus HKTs in a recirculating flume, finding that wake evolution was more affected by the turbines' rotational

direction than their spacing. Slower wake recovery occurred when blades counter-rotated with the inflow, while faster recovery was observed when counter-rotating against it.

More recently, other wind tunnel investigations were carried out for VAWTs in close-proximity. For instance, Talamalek et al. [21], using wind tunnel tests, compared the wake of counter-rotating VAWTs to a stand-alone turbine. They reported that paired turbines showed a larger velocity deficit and wider wake area, with minimal differences in overall wake characteristics. Paired VAWTs shifted momentum transport from lateral to vertical advection, whereas isolated VAWTs exhibited higher lateral turbulent transport and slightly better wake recovery. Jodai et al. [22] investigated pairs of VAWTs in both in-line and staggered configurations, finding that the in-line setup is more efficient.

2.1.2 Numerical Investigations

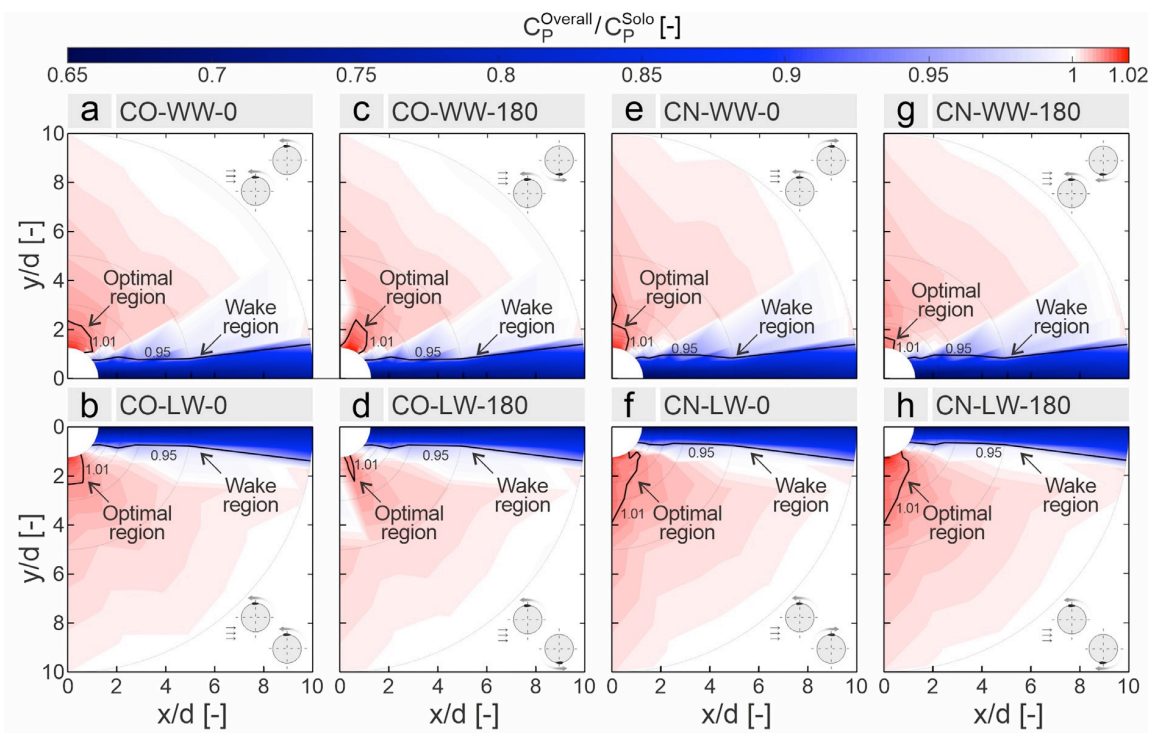


Figure 2.2: Performance mapping for the influence of cluster layout in different configurations of arrays of two turbines (adopted from [23]).

In addition to experimental studies, numerical investigations, in particular two-dimensional CFD, offer greater flexibility and reliability for testing multiple designs and configurations, enabling deeper analysis.

By the means of two-dimensional uRANS simulations, several studies investigated optimisation strategies for rotor arrangements. Mohamed et al. [24] explored various in-line and staggered layouts, concluding that in-line configurations generally enhance efficiency better than staggered ones. This finding was further confirmed by extensive analyses for co-rotating pairs (Sahebzadeh et al. [16]) and counter-rotating pairs (Sahebzadeh et al. [23]), concluding that staggered layouts could be more efficient only if the oblique angle (angle between the adjacent turbines) is an in-line angle. As can be seen in Figure 2.2, a detailed map is showing the performance enhancement/degradation for a Darrieus turbine pair based on the position of the second rotor

reference to the first one, showing that almost for all cases, the optimum oblique angle is around 75° to 90° . Zheng et al. [25] also confirmed the superior performance of in-line setups, recommending a spacing of $1.5D$ to $1.6D$ for maximizing cluster efficiency.

Also using 2D CFD, Guilbot et al. [26] examined the performance of twin two-bladed Darrieus rotors, investigating how rotor spacing and rotational orientation affected efficiency. Peng et al. [27], using the Taguchi method, analysed the influence of aerofoil shape, pitch angle, solidity ratio, rotational direction, and spacing on the optimal tip-speed ratio of closely spaced turbines. They identified solidity ratio as the most influential factor, consistent with findings for single rotors [28], but noted that twin rotors have a higher optimal TSR than single rotors.

CFD studies have also been used to understand the physical mechanisms behind power augmentation in twin Darrieus rotors. Through 2D uRANS simulations, Zanforlin et al. [29] proposed two reasons for power enhancement: a favourable deviation in absolute wind velocity and wake contraction in the downwind path, which increases available momentum flux. Alexander et al. [30] expanded on this, noting that closely deployed rotors increase the volumetric flow entering the turbines due to the blockage effect. They also observed that most power coefficient gains occur during the mutual interaction zone, while performance may decrease in other azimuthal positions. Zhao et al. [31] conducted a similar three-dimensional analysis, incorporating a deflector between the turbines. They found that the deflector improved performance, increasing the torque coefficient by 23% in the optimal configuration, but it also delayed wake recovery due to blocked high-speed flow and stronger tip vortices.

2.1.3 Discussion

The concept of closely deploying Darrieus turbines has recently gained attraction in the industrial sector, exemplified by applications such as Nenuphar's Twinfloat concept [32], which features two floating Darrieus wind turbines on a shared platform, and HE-Powergreen's array [33], comprising four hydrokinetic Darrieus turbines mounted on a single supporting structure in a channel. However, in wind energy applications, this compact arrangement has a key drawback: increased sensitivity to inflow direction, which undermines the omnidirectional advantage of Darrieus turbines. For HKTs, where water streams are unidirectional, closely spaced configurations provide an optimal solution for power generation.

Based on the reviewed literature, previous works can be categorised into three main areas: i) layout optimization studies, ii) wake studies, and iii) studies focusing solely on the physics involved when turbines are deployed in close proximity. It is important to note that the majority of the reviewed works focused on wind applications, while studies on HKTs often neglect the effect of the two-way interactions between the array and the channel, which could impact the optimisation of the array configuration.

Table 2.1 summarises the key studies on clustering Darrieus turbines in recent years. However, despite these contributions, there is a lack of quantitative analysis on the kinematic behaviour of turbines within the mutual interaction region, which could provide valuable insights into maximizing the potential of this compact configuration. Specifically, most existing studies concentrate on optimizing the overall power coefficient, with limited focus on the underlying physical mechanisms, often relying on qualitative analysis (e.g., Refs. [26,29,30]). In other words, while previous works have thoroughly analysed flow fields, none have systematically investigated how mutual flow interactions in the small gap between turbines affect the local absolute inflow to the blades, thus contributing to efficiency improvements. Advancing the concept of closely

spaced Darrieus turbines requires a detailed understanding of local flow kinematics, which is critical for optimizing rotor design aspects such as aerofoil selection and pitch control. Furthermore, this knowledge is essential for validating and improving low- and medium-fidelity simulation tools used to model Darrieus turbine performance.

Table 2.1: Summary of the studies on deploying Darrieus turbines in close proximity.

Reference	Year	Method	Main objective	Main conclusion
Dabiri [14]	2010	Experimental	Investigate the use of counter-rotating VAWTs	The power coefficient of turbines in array can exceed that of the stand-alone by $\approx 10\%$
Whittlesey et al. [34]	2010	Experimental and numerical	Development of Semi-empirical potential flow model for an array of VAWTs	VAWT arrays inspired by fish schooling achieve compact layouts with power density over ten times higher than HAWT farms.
Kinzel et al. [17]	2012	Experimental	Analysis of the flow field along 9 counter-rotating pairs of VAWTs	Pairing VAWTs reduces the flow recovery distance to 6 D, significantly less than the 14 D required for HAWTs, enabling closer turbine spacing in wind farms.
Zanforlin et al. [29]	2016	Numerical (2D CFD)	Identify the key flow mechanisms contributing to the enhancement of the pairs of counter-rotating VAWTs	In side-by-side turbine arrays, the neighbouring turbine alters lateral velocity, making the flow direction more favourable for lift and torque generation. Staggered turbines cannot outperform in-line layouts, with local flow mechanisms being crucial for small VAWT farms.
Mohamed et al. [24]	2019	Numerical (2D CFD)	Investigate the effect of turbines spacing and oblique angle on two and three closely spaced VAWTs	The optimum configuration is that of in-line arrangement with oblique angle of 0° with a 23% increase in power coefficient compared to the stand-alone turbine.
Alexander et al. [30]	2020	Numerical (2D CFD)	Examine the physical mechanism related to the performance improvement in closely spaced VAWTs	The 10% improvement in power coefficient for double-rotor VAWTs is mainly due to reduced bypass flow, as the second rotor suppresses cross-stream flow, redirecting more flow through the rotors and lowering interior bypass, despite an increase in exterior bypass.

Vergaerde et al. [18]	2020	Experimental	Wind tunnel tests for the performance leeward and windward counter-rotating VAWTs	Wind tunnel tests show paired turbines achieve a 13–16% higher power coefficient than two individual turbines combined, aligning with numerical studies.
Peng et al. [27]	2020	Numerical (2D CFD)	Investigate the influence of aerofoil section, solidity ratio, pitch angle, rotational direction, and turbine spacing on the performance of twin VAWTs	The relative importance of the evaluated parameters are: Solidity > pitch angle > aerofoil section > rotational direction > turbine spacing. The increase in power coefficient observed for all evaluated cases ranged between 8 – 13%.
Guilbot et al. [26]	2020	Numerical (2D CFD)	Study the aerodynamic interactions occurring in a pair of VAWTs	The power enhancement in double-rotor VAWTs results from extended lift generation in the upwind path, influenced by counter-rotating directions, with confined cross-stream velocity the increases the angle of attack, lift, and torque.
Zheng et al. [25]	2020	Numerical (2D CFD)	Optimise the efficiency based on the inter-spacing in clusters of two and three turbines	Lateral constraints and upstream blockage significantly impact power generation, with 3-VAWT clusters generally outperforming 2-VAWT setups; the optimal 3-VAWT arrangement achieves an 11.1% higher average power coefficient than a single turbine.
Sahebzadeh et al. [16]	2020	Numerical (2D CFD)	Investigate the effect of relative spacing and oblique angle in a pair of co-rotating VAWTs	In co-rotating pairs, the maximum performance is achieved in staggered configuration with the minimal distance between the turbines and an oblique angle of about 45°.
Li et al. [19]	2021	Experimental	Investigate the effect of solidity and other parameters on the performance of a counter-rotating VAWTs pair in urban environments.	Closely-spaced VAWTs show minimal power output differences under low urban wind speeds but improve slightly with higher rotor solidity.

Zhao et al. [31]	2021	Numerical (2.5D CFD)	Investigation of the performance of a pair of VAWTs with a deflector in between the rotors	The deflector might increase the favourable angle of attack during the areas of mutual interaction. Under the optimum conditions, the pair with deflector can achieve about 30% more power coefficient compared to the case without deflector.
Müller et al. [20]	2021	Experimental	Characterisation of the wake developed downstream a stand-alone turbine and a twin Darrieus VAWTs at different spacings	Wake dynamics are more influenced by turbine rotational direction than lateral spacing, with counter-rotating forward setups showing faster wake recovery, while same-direction setups yield greater lateral expansion and reduced velocities.
Sahebzadeh et al. [23]	2022	Numerical (2D CFD)	Create a performance map on the impact of turbine relative arrangement in VAWT pairs in all configurations	The performance of VAWT arrays varies significantly with rotational direction and positioning, $\approx 8\%$ efficiency increase with counter-rotating leeward setups achieving the highest efficiency due to favourable wake interactions, while windward setups perform best when co-rotating.
Talamalek et al. [21]	2024	Experimental	Investigate the wake replenishment mechanisms behind paired VAWTs	Paired VAWTs exhibit minimal differences in overall wake recovery compared to isolated ones, with enhanced vertical advection balancing the increased lateral advection seen in isolated setups.
Jodai et al. [22]	2024	Experimental	Investigate different arrangements of closely spaced VAWTs: in-line pair, staggered pair, and in-line three turbines	Reducing the inter-spacing improves efficiency, with in-line setup achieving up to a 6% surplus in efficiency, staggered pair shows a power drop for downstream turbines.

2.2 Modelling of Darrieus Turbines

There are several methods available in the literature to simulate the performance of Darrieus turbines. This section reviews various modelling techniques, focusing on the basic formulation of each method, computational power required, and. Thereafter, a discussion is presented highlighting the differences between modelling techniques in terms of accuracy computational

cost in the context of identifying the better candidate for modelling Darrieus turbines for hydrokinetic applications.

2.2.1 Double-Multiple Streamtube (DMST)

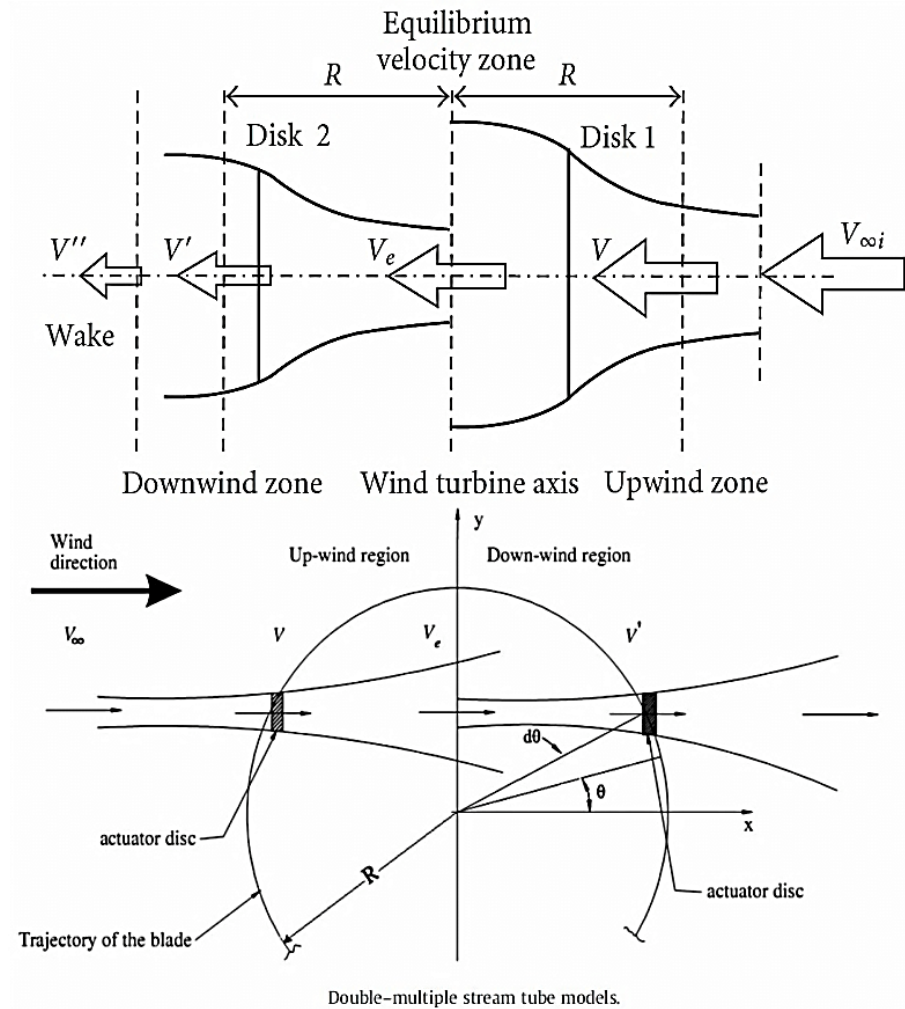


Figure 2.3: Illustration of the Double-multiple Streamtube (DMST) model (adopted from [36]).

The concept of representing a VAT rotor as two aerodynamically independent rotor halves in series dates back to Lapin [35] in 1975. Building on this idea, Paraschivoiu [36] proposed the Double-Multiple Streamtube (DMST) model as an analogy to the actuator disk concept. The DMST represents the VAT rotor as a series of adjacent streamtubes, each containing two actuator disks, as illustrated in Figure 2.3. It is an analytical model that divides the rotor into two parts: upstream and downstream halves. The induced velocity is computed separately for each part, enabling the calculation of lift and drag forces on each disk and, subsequently, the overall rotor performance. The model was later refined [37] to account for the azimuthal variation of induced velocities in the upwind and downwind disks. In a DMST-based model, Cacciali et al. [38,39] introduced an iterative algorithm to adjust the upstream inflow of a Darrieus hydrokinetic turbine based on downstream conditions in a free-surface channel.

Despite the low computational cost of the DMST model, it inherently neglects significant aspects of VAT rotors that become critical depending on rotor geometry and operating conditions. These include streamtube expansion and blade-wake interactions, limiting the ability of the DMST models to estimate local performance and confining its accuracy to global performance [40]. Furthermore, DMST models are unsuitable for turbines with high solidity and tip-speed ratios due to the correlation between rotor loading and the induction factors calculated for each streamtube. At high induction factors, the underlying momentum theory breaks down, resulting in nonphysical predictions of wake velocity and drag [41]. Such inconsistencies are addressed in HAT codes based on Blade Element Momentum (BEM) theory by applying empirical corrections for drag. However, these corrections are not directly applicable to VAT codes, as wake flow quantities must also be corrected [42].

2.2.2 Actuator Cylinder (AC)

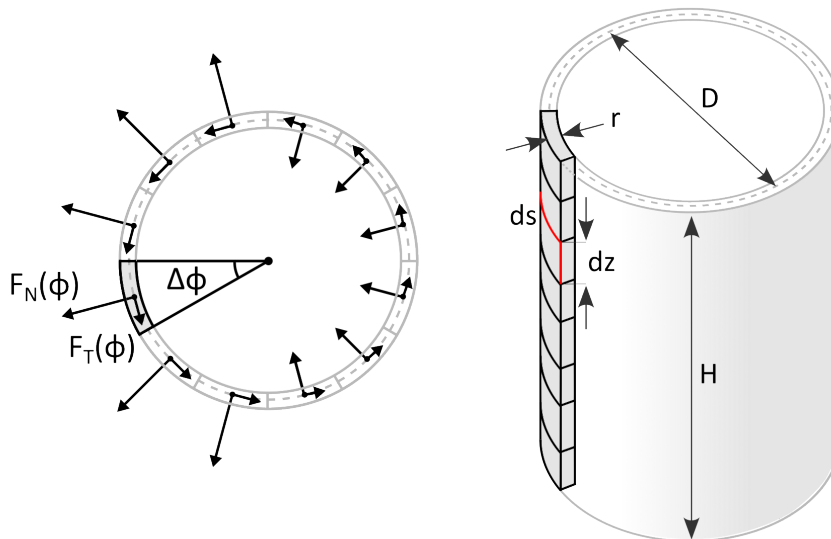


Figure 2.4: Actuator cylinder representation of VAT rotors.

The Actuator Cylinder (AC) model was proposed by Madsen [43] to represent an ideal VAT rotor as an extension of the actuator disk method commonly used for HATs. Due to the fundamental differences between the swept areas of HATs and VATs, the AC model was introduced as a basis for computing the ideal power coefficient, analogous to Betz's law for HAT rotors. The initial AC model was semi-analytical, representing the turbine rotor as volumetric forces distributed along a cylinder, aiming to resolve the flow field influenced by these forces. Madsen et al. [44] later expanded on this concept by integrating the AC model into an aeroelastic code to simulate the performance of a floating vertical-axis wind turbine.

Shamsoddin and Porté-Agel [45] applied the AC concept, referred to as the actuator swept surface model, within an LES (Large Eddy Simulations) framework alongside the ALM. They implemented the actuator swept surface by distributing time-averaged forces along the cylinder, which were then projected onto the computational domain as volumetric forces, similar to the conventional implementation of the actuator disk for HATs. Ning [46] evaluated the capability of the AC to predict power augmentation in closely spaced Darrieus turbines, emphasizing the need

for further refinements and detailed validations in this context. Shives et al. [47] investigated wake interactions and channel blockage effects in HKT arrays, though their analysis did not consider the impact of the free water surface.

Despite the extensive studies and various implementations of the AC model, no studies in the literature have validated its application under realistic conditions for hydrokinetic systems. Specifically, its capability to capture interactions between turbines within an array, as well as interactions between the turbines and the water resource, remains unexplored.

2.2.3 Lifting Line Theory (LLT)

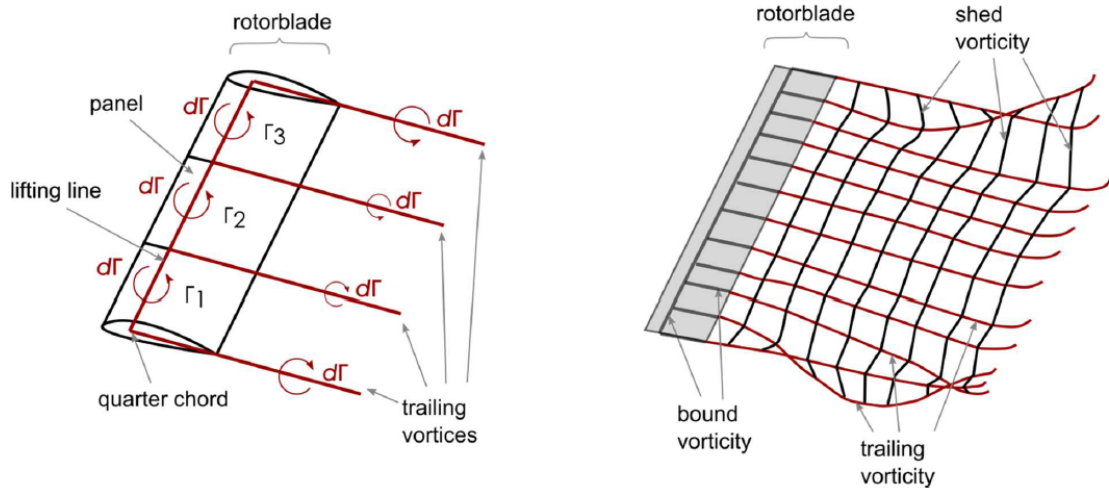


Figure 2.5: Illustration of the Lifting Line Theory

Models based on Lifting Line Theory (LLT), also referred to as vortex models, are commonly used to simulate lifting bodies. Unlike other models, LLT-based approaches rely on the conservation of vorticity rather than mass and momentum. This family of models are considered as a compromise between the computationally intensive CFD-based models and the models based on the BEM, known also as engineering models.

As shown in Figure 2.5, in the vortex-based models, the blade is modelled as a lifting line, discretised into individual vortex filaments, with the lifting line representing the hydro/aerodynamic centre. The lift and drag forces, computed using the Kutta-Joukowski theorem, determine the strength of each vortex source. Kelvin's circulation theorem, which governs the conservation of circulation, is applied to evaluate the impact of fluid-structure interactions imposed by the turbine blade on the flow field, as expressed in Eq. 2.1:

$$\frac{d\Gamma}{dt} = \frac{d\Gamma_{blade}}{dt} + \frac{d\Gamma_{wake}}{dt} = 0 \quad 2.1$$

According to Eq. 2.1, temporal variations in the blade aerodynamic loads result in the release of two types of free vortex filaments into the wake: trailing and shed vortices (illustrated in Figure 2.5). The interaction between these vortex systems alters the velocity field, with the resulting deformation described by the Biot-Savart law. By incorporating this phenomenon along with the turbine's kinematic and geometric parameters, one can compute the average relative flow velocity and angle of attack for the aerofoil. These values are then used to estimate the

aerodynamic forces, based on pre-tabulated lift and drag coefficients. This process is inherently iterative, requiring successive refinements to achieve convergence.

One of the most widely known LLT-based codes for VATs is the one introduced by Murray and Barone [48], implemented in CACTUS. Similarly, Marten et al. [49] implemented their code in QBlade, and Shaler et al. [50] implemented theirs in OLAF. Branlard et al. [51] built upon the OLAF code and compared it against other models, including DMST-based code and CACTUS, as well as experimental measurements. They concluded that blade-vortex interaction is not well reproduced when compared to experimental measurements. They also pointed out that different lifting-line vortex code implementations, such as OLAF and CACTUS, yield varying loads and induction fields due to differences in trailing-edge vortex handling, control point placement, and angle of attack calculations. De Tavernier et al. [52] used a two-dimensional vortex model to investigate the performance of VAT pairs with different layouts, demonstrating that LLT-based models are capable of reproducing the power augmentation effect observed in Darrieus turbine clusters.

2.2.4 Actuator Line Method (ALM)

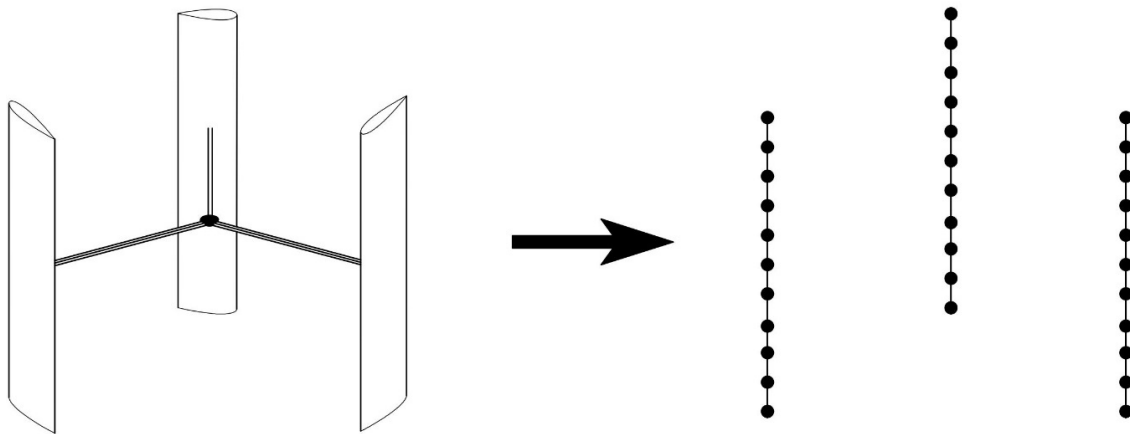


Figure 2.6: Illustration of the Actuator Line Method.

The Actuator Line Method (ALM) is a hybrid approach initially developed for simulating HATs [53]. In this method, the turbine blade is not explicitly resolved but is instead modelled using a lumped parameter approach. As illustrated in Figure 2.6, the blade is represented as an actuator line, where the local flow velocity around each actuator line is sampled. Based on this sampled velocity, the local kinematic variation around the hypothetical blade is calculated. Using these kinematic data, equivalent forces coefficients are retrieved from tabulated polar data, and the resulting forces are projected into the CFD flow field as a volumetric flow source. Since the geometry of the turbine blade is not explicitly resolved, the overall grid density is significantly reduced, resulting in a corresponding decrease in computational cost with respect to blade-resolved CFD. For HATs, the ALM has reached a satisfactory level of maturity [54].

Shamsoddin and Porté-Agel [45] were the first to apply the ALM to study the wakes of VAWTs using LES, demonstrating good agreement with experimental flow field measurements. They subsequently conducted another study to investigate the effect of the atmospheric boundary layer on VAWT wakes [55]. Abkar and Dabiri [56] used the ALM to characterise VAWT wakes and

developed an analytical Gaussian wake model for the estimation of VAWT wakes. Mendoza et al. [57] applied the ALM coupled with LES to study the near wake of a VAWT, showing strong agreement when comparing wake results to PIV measurements.

Despite the application of the ALM to study VATs, none of the earlier studies properly validated its capability to estimate turbine performance. To address this, Melani et al. [58] recently developed and validated an ALM code specifically tailored for VATs by incorporating additional features and sub-models to account for the unique characteristics of VAT performance. As detailed in Section 1.2, VATs are characterised by unsteady and complex dynamics, unlike the HATs for which the ALM was originally designed. These enhancements included improvements in velocity sampling through the integration of the Line-Average method [59]. Additional sub-models were also introduced to capture effects specific to Darrieus rotors, such as the blade curvilinear effect [60], the influence of pitching moments on torque [61], and dynamic stall [62]. These recent developments open the door for further developments in more advanced models that incorporate VATs, making the design process more efficient and accessible. It is important to note that the latter ALM code has been further developed and used in this thesis.

2.2.5 Computational Fluid Dynamics (CFD)

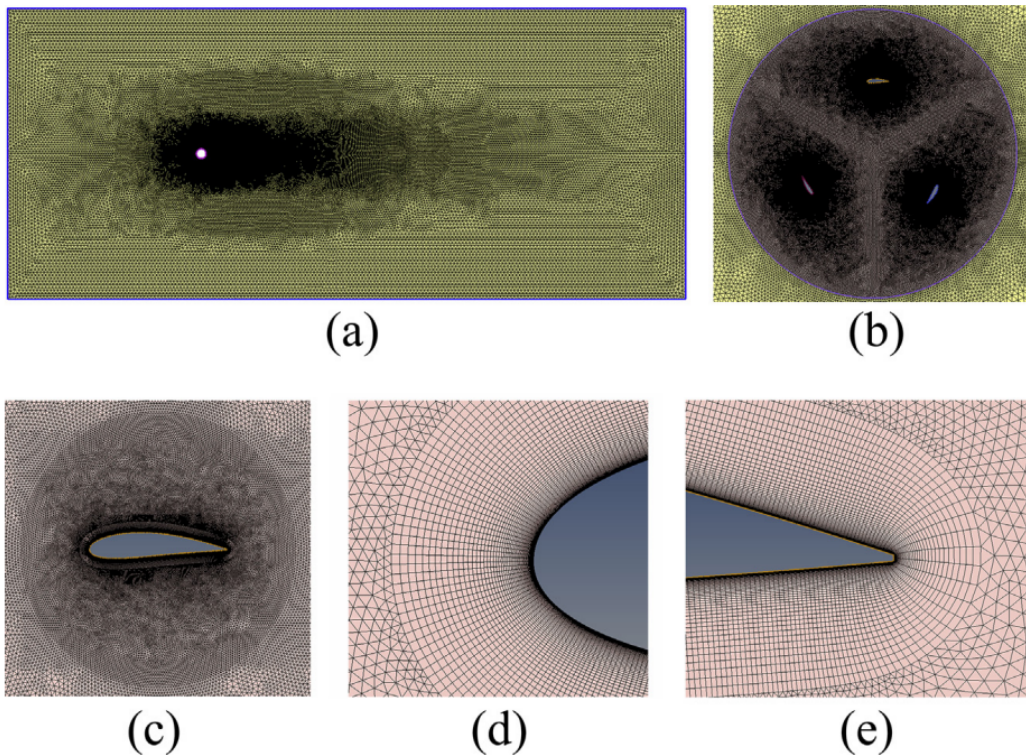


Figure 2.7: Example of the mesh details required for a proper blade-resolved CFD for a Darrieus turbine, (a) full domain, (b) rotor, (c) aerofoil, (d) leading edge, and (e) trailing edge. Adopted from [63]

As discussed in Section 1.2, the performance of Darrieus turbines is inherently characterised by unsteadiness due to continuous variations in the flow kinematics local to the blade, particularly the angle of attack. To capture the effects of this unsteadiness on load and flow fields, CFD is commonly employed, with unsteady Reynolds-Averaged Navier-Stokes (uRANS) solvers

widely used to study the aerodynamic and hydrodynamic characteristics of these machines [64–66]. In general, CFD methods vary in their approach, but the most common is known as blade-resolved CFD, where the blade geometry is explicitly resolved up to the boundary layer. Figure 2.7 illustrates an example of meshing around a Darrieus turbine blade. The highly refined mesh demonstrates the level of detail required, highlighting why such analyses are computationally expensive. Moreover, while LES offer superior accuracy, their computational cost is prohibitively high, limiting their use in blade-resolved analyses of Darrieus turbines to a few studies.

Balduzzi [63] conducted a comprehensive analysis of the critical factors to consider when modeling Darrieus turbines using blade-resolved CFD. These factors include the turbulence model, convergence criteria, domain dimensions, mesh size, and temporal discretization. Neglecting these considerations can lead to misleading estimations but addressing them significantly increases computational time. As a result, most studies on Darrieus turbines are restricted to two-dimensional domains, particularly when investigating multiple turbines (see Section 2.1.3).

In a dedicated study, Franchina et al. [67] compared the performance of 2D and 3D uRANS models against experimental measurements, demonstrating that the 3D model imposes fewer restrictions due to the fully three-dimensional nature of the flow around the turbine. More recently, He et al. [68] assessed the accuracy of various blade-resolved CFD approaches, including 2D uRANS, 2.5D uRANS, 2.5D LES, 3D uRANS, and 3D LES. The 2.5D method differs from both 2D and 3D by modelling a limited blade height with periodic conditions at the top and bottom boundaries. Their study concluded that 2D uRANS significantly overestimates power coefficients due to limitations in vorticity diffusion modelling, whereas 2.5D LES, 3D LES, and 3D uRANS provide much more accurate predictions compared to experimental data.

Despite its accuracy, three-dimensional CFD is unfeasible at the industrial level, particularly in the context of arrays of HKTs. This complexity arises due to the need to model not only individual turbine blades but also the interactions between turbines in arrays and their surroundings, such as channel walls or the free water surface. While two-dimensional approaches cannot adequately capture the full scope of these interactions, three-dimensional CFD remains computationally prohibitive. To address this gap, lower-fidelity hybrid methods have been developed. These approaches use CFD to estimate the flow field around the turbines while employing simplified turbine force models to reduce computational effort.

2.2.6 Discussion

Table 2.2 summarises the main comparative studies found in the literature assessing different modelling techniques along with their main conclusions. In general, it can be said that the modelling techniques of Darrieus turbine vary significantly in terms of their capabilities to capture complex phenomenon which is always directly related to the computational time needed to perform the calculation; methods that include CFD would be computationally more expensive than other analytical models, however, more accurate.

In a comparison between the ALM and AC within a framework of LES, Shamsoddin and Porté-Agel [44] found that the ALM produced more reliable results compared to the AC when validated against experimental measurements. This superiority was attributed to the ability of the ALM to resolve the periodic location of blade forces, enabling the modelling of the unsteady periodic nature of the wake. However, they identified the AC as a promising tool for modelling groups of VATs. In another dedicated comparison, Ferreira et al. [69] evaluated the DMST and AC based

models, along with others, demonstrating that the DMST model is inconsistent across various operating and geometric conditions.

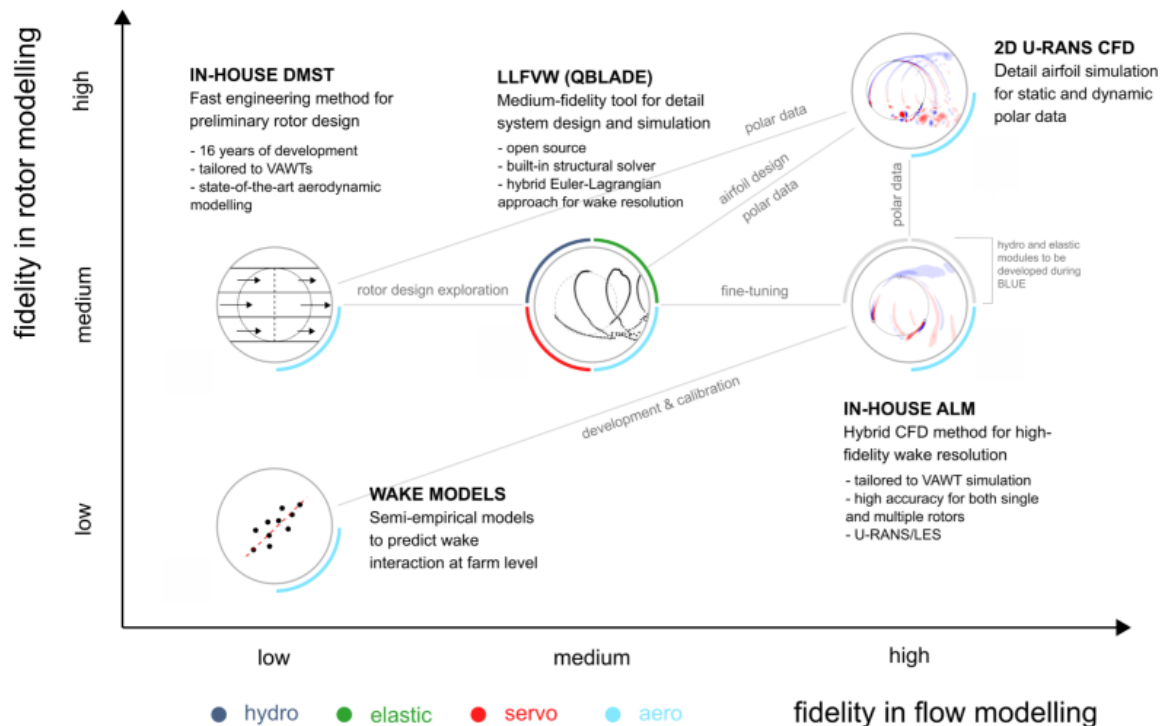


Figure 2.8: Overview on the numerical models used to simulate the performance of Darrieus turbines along with their fidelity level in modelling rotor performance and surrounding flow field.

Figure 2.8 qualitatively illustrates the difference between the modelling techniques commonly used to simulate the performance of Darrieus turbines. It shows the fidelity level of each model in predicting either rotor performance or the flow field. The 3D blade-resolved uRANS is not considered here due to its impracticality, particularly at the industrial level. It is important to note that, in the case of HKTs, the predictions of the flow field are crucial for estimating rotor performance, as both are interdependent and influence each other.

Based on the reviewed literature, it is important to note that the interactions among Darrieus turbines within an array, as well as the interaction of the array with its surrounding environment, such as an open channel in the hydrokinetic case, can only be effectively studied using CFD-based methods. This approach is necessary to capture the complexities involved and their impact on both turbine performance and the surrounding flow field.

Table 2.2: Summary of the comparative studies assessing different modelling methods for VATs.

Reference	Year	Main objective	Main conclusion
Shamsoddin and Porté-Agel [45]	2014	Simulating the turbulent wake behind VAWT using ALM and AC	The ALM surpasses the AC in capturing VAWT wake dynamics, blade-wake interactions, and turbulence intensity, offering better alignment with flow physics and wake recovery predictions. However, the AC remains advantageous for wind farm simulations due to its lower grid resolution requirements.

Ferreira et al. [69]	2014	Comparison between six models: including two variations of DMST, three variation of the vortex methods and the AC	The models based on DMST show significant deviation due to its fundamental flaw in predicting pitch angle effects, limiting its reliability unless empirically corrected. In contrast, vortex panel methods (U2DiVA, CACTUS, ARDEMA2D) and the AC approach align well with each others. However, discrepancies in angle-of-attack estimates among these models raise concerns about their design-level accuracy. Further research is needed to explore their behaviour under gusts and their implications for fatigue life estimation.
Cheng et al. [70]	2016	Comparison between a developed AC and DMST models in modelling floating VAWTs	the developed AC model provides more accurate predictions of aerodynamic power and loads compared to the DMST method.
Delafin et al. [71]	2017	Comparison between DMST, free-wake vortex model, and 3D blade-resolved uRANS in modelling Sandia 17 m VAWT	DMST models are prone to inaccuracies in predicting optimal tip speed ratios and forces, with limited applicability to new designs due to reliance on calibrated parameters. Free-wake vortex models provide better accuracy and balance computational cost. CFD delivers the most precise results, especially at higher tip speed ratios, but is computationally expensive and less reliable at low tip speed ratios.
Franchina et al. [67]	2019	Comparison between 2D and 3D blade-resolved uRANS for a turbine with aspect ratio of ≈ 1.5	Results from 2D uRANS do not align well with its corresponding 3D uRANS, as the fully 3D flow around VAWTs significantly affects midspan flow distribution.
De Tavernier et al. [72]	2020	Comparison between four models in modelling the H-rotor and ϕ -rotor (H-Darries and Darries in Figure 1.1): variations of AC (2D), vortex panel method, and ALM	While AC models lack 3D representation, and models like free-, fixed-vortex wake and ALM incorporate tip corrections but face physical or computational limitations, the vortex models, particularly with fewer assumptions, provide the most reliable results.
He et al. [68]	2020	Comparison between 3D, 2.5D, 2D uRANS and LES	2D uRANS struggles to accurately predict VAWT aerodynamic

		blade-resolved CFD in modelling VAWTs	characteristics, often attributed to ignoring blade tip vortices. However, 2.5D LES results suggest this is not the primary issue. Instead, uRANS overestimates tangential forces and delays dynamic stall, unlike LES, which better captures unsteady flows.
Ajay et al. [73]	2024	Comparison between six different models for the modelling of the X-rotor concept: a DMST model, a 2D AC, two variations of the vortex panel method, a new method based on the Lattice Boltzman Method (PFlow), and a 3D blade-resolved uRANS	The DMST model deviated significantly from other predictions, while the AC model was consistent at small pitch offsets but struggled with blade-vortex interactions due to its quasi-steady, 2D nature. QBlade and CACTUS showed similar results, with QBlade slightly overpredicting power. URANS and PFlow aligned well with vortex models for thrust but showed lower power due to blockage effects and tower modeling, while reduced forces on the turbine's downwind half resulted from blade flow separation and CFD flow curvature effects.

2.3 Darrieus turbines in hydrokinetic energy

This section provides a review of the literature on hydrokinetic turbine modelling and the constraints that should be considered when simulating HKTs under realistic conditions, with emphasise on the Darrieus HKTs. The section begins with a description of the interaction between the water resource and HKTs, highlighting its relevance and importance. It then discusses the Volume of Fluid (VOF) method, a commonly used technique in the literature to simulate the effects of the free water surface in HKT energy systems, with a focus on studies related to Darrieus turbines.

2.3.1 HKT-Channel Two-way Interactions

For accurate modelling of HKTs, particularly in arrays where high blockage is significant, careful attention must be paid to the interactions between the deployed turbines and the resource, in the present case, a small water course. Garrett and Cummins [74] noted that the maximum power output of HKTs is constrained by the blockage effect, as excessive turbines obstruct the flow and reduce efficiency. Similarly, Nishino and Willden [75] emphasised that blockage increases when a turbine array is deployed rather than a standalone turbine. In this scenario, blockage can be categorised as either global or local. Global blockage refers to interactions between the turbines and the channel resource, while local blockage pertains to interactions among turbines within the array (see section 2.1). Consequently, the design of the array layout, including turbine spacing and relative positioning, becomes crucial to optimizing HKT performance [76].

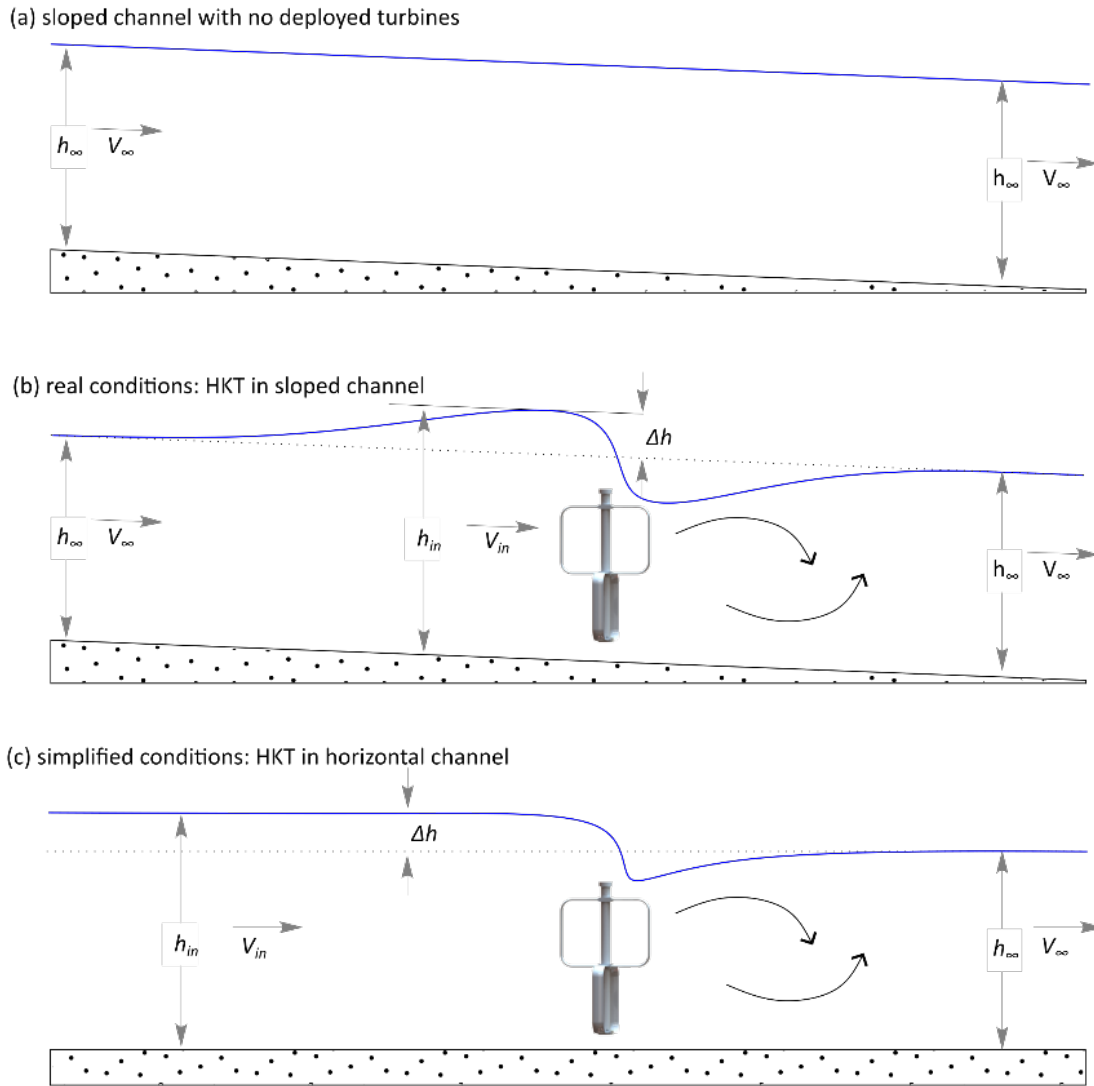


Figure 2.9: Schematic representations of (a) sloped bed channel without turbine, (b) with turbine, and (c) horizontal bed channel with a turbine.

Kirke [77] criticised what he called “the zero head myth”, which is the common belief among the research community that HKT does not result in a drop in the ambient water pressure, in another word, the water level. Arguing that in hypothetical uniform and very long channel, there must be a global decrease in the water level between far upstream and far downstream the deployed HKTs in order to achieve the energy balance between the friction with the channel walls and turbulence around obstacles. However, this head drop might be very small and even unnoticeable. Indeed, this analysis even relevant, but it is not significant in analysing the HKT performance, where what is more relevant is the channel hydraulic performance close to the turbines as analysed in details by Kinsey and Dumas [78].

As depicted in Figure 2.9-a, for a sloped channel with a uniform cross-section, a steady equilibrium is maintained between the gravitational force driving the flow due to the channel slope and the friction along the channel walls. Under these conditions, the inlet water depth h_∞ and flow velocity V_∞ remain constant as long as the flow remains steady and uniform. Introducing HKTs into this system (Figure 2.9-b) disrupts the equilibrium, causing changes in the water level

and flow velocity upstream of the turbine. The blockage effect induced by the turbine raises the upstream water level to a maximum height h_{in} while reducing the flow velocity to V_{in} . Downstream, the water level drops before eventually stabilizing back to its original value h_{∞} , or slightly less, as noted by Kirk [77], a process known as "depth recuperation". The upstream rise in water level corresponds to a decrease in hydraulic head, quantified as $\Delta h = h_{in} - h_{\infty}$, which is a valid assumption when head dissipation is negligible due to its small value. Here, the determination of Δh is crucial to separate the influence of channel blockage on turbine performance. Otherwise, the computed power coefficient would be misleading, as the reduction in flow upstream of the turbines due to this blockage would make the efficiency appear higher than it actually is [79]. The correction power coefficient $C_{P,corr}$ based on the channel blockage can be computed as in Eq. 2.2:

$$C_{P,corr} = C_P \left(1 + \frac{\Delta h}{h_{\infty}} \right)^{-3} \quad 2.2$$

For instance, an experimental campaign has been carried out by Mayers and Bahaj [80] on a HKT operating in a recirculating channel. They observed a rise in water level relative to the free surface near the upstream due to blockage, a decrease near the downstream due to energy extraction, followed by a subsequent rebound in the far downstream due to wake recovery. These observations have been confirmed in other studies [81].

Resolving such a system using CFD would necessitate the use of very long computational domains, as highlighted by Gauvin-Tremblay and Dumas [79], which might be computationally infeasible in many cases. An alternative approach involves employing a simplified channel that assumes a horizontal channel bed (referred to as the "horizontal bed assumption"), resulting in a water level profile similar to that illustrated in Figure 2.8-c. While this assumption simplifies the model by eliminating the need to account for bed slope variations, it limits the analysis by preventing the calculation of the distance over which depth recuperation occurs. However, it still allows for the estimation of the upstream water level rise, which is the most important factor in evaluating HKT performance.

In order to account for the changes in the free water surface, thus, considering the blockage effect due to the presence of HKTs, several methods could be used including analytical models and models based on CFD. For example, Cacciali et al. [38,39] developed a DMST-based model that iteratively updates the inflow upstream to a Darrieus HKT based on the downstream conditions in a free surface channel environment.

2.3.2 Volume of Fluid (VOF)

The Volume of Fluid (VOF) method, developed by Hirt and Nichols [83], is a numerical technique commonly used in CFD simulations to model free surface flow, where the interface between different fluids (e.g., water and air) is dynamic and needs to be tracked. The method works by solving the flow fields of water and air using a single set of momentum equations while tracking the volume fraction of each fluid across the domain, allowing the computation of the two-phase solution. It is based on the principle that two or more fluids cannot penetrate each other. The volume fraction of each fluid is introduced as a new variable for every computational cell. The volume fraction equation is then solved for the secondary phase, while the primary phase is computed in such a way that the total volume fraction sum is unity [84].

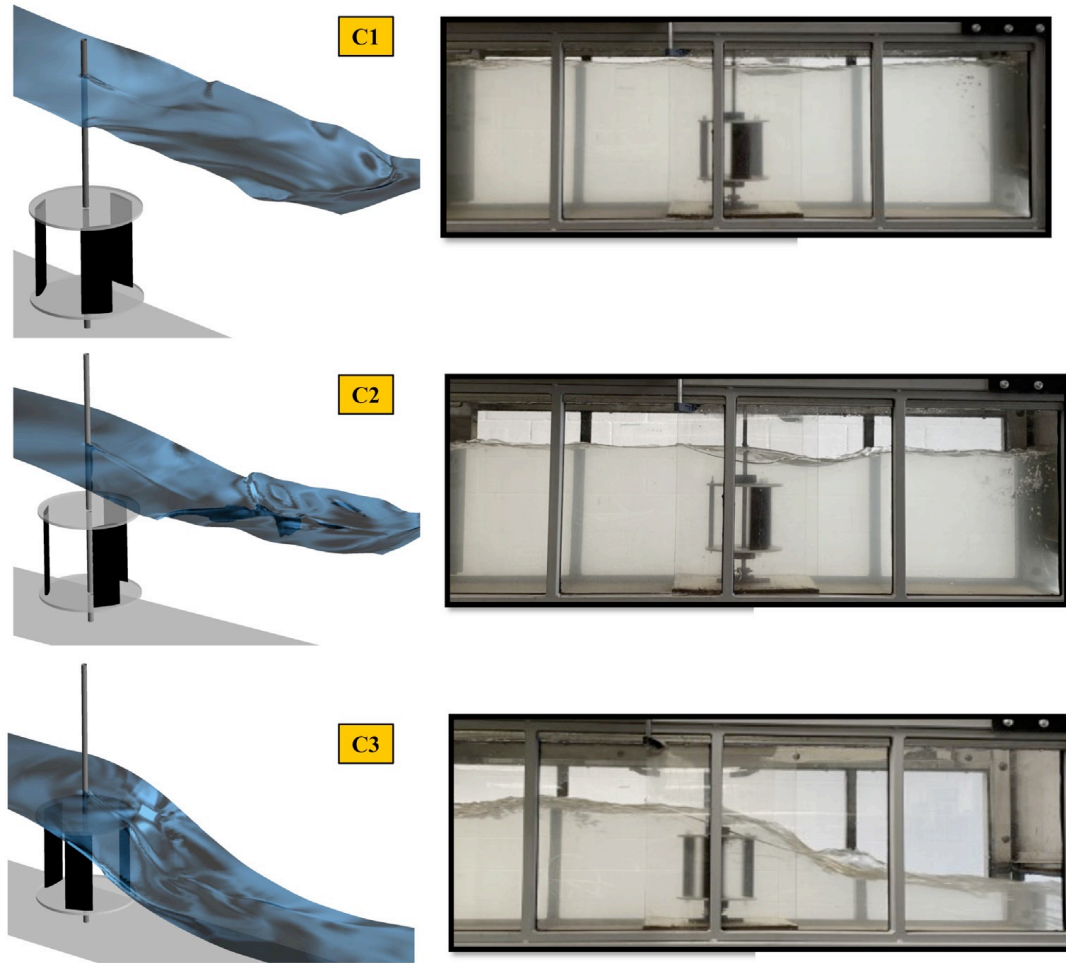


Figure 2.10: Free surface variation simulated using VOF compared to corresponding experimental results (adopted from [82])

Several studies in the literature have used the VOF approach to simulate the performance of horizontal axis HKTs [85–88]. However, similar studies on VATs are rare, and when found, they often focus on a single stand-alone turbine. For example, Gauvin-Tremblay and Dumas [79] used VOF and blade-resolved CFD simulations for a Darrieus turbine and found that immersion depth does not significantly affect power extraction, making it independent of the Froude number when the free water surface is considered. More recently, Yosry et al. [82] conducted blade-resolved CFD studies for a Darrieus HKT using the VOF method and validated the numerical model against water tunnel experiments. Figure 2.10 illustrates a free water surface modelled using VOF compared to experimental results [82].

It is important to note that other models exist to model the free water surface in the context of hydrokinetic turbines, either analytical models or those who utilise CFD. For example that of Apsley and Hu [89], known as moving-mesh surface-fitting approach, used by Apsley et al. [90] to simulate an array of horizontal axis HKT using ALM.

2.3.3 Discussion

Given the previous review, it is evident that few studies focus on simulating the performance of Darrieus turbines for hydrokinetic applications under realistic conditions, particularly those that include free water surface modelling. Moreover, it is important to underline that, as per the author's knowledge, no studies have addressed aggregate blockage effects, such as channel-to-turbine and turbine-to-turbine interactions. While the latter can be modelled as outlined in Section 2.1, the VOF method, as detailed in [79,82], when coupled with blade-resolved CFD, has proven effective in simulating free water surfaces and addressing uncertainties in modelling channel-to-turbine interactions.

Although Cacciali et al. [38,39] developed models that incorporate both blockage effects (as described in Section 2.2), these analytical models are primarily used as rapid tools for preliminary design, given that BEM-based models in VATs cannot accurately predict second-order phenomena, such as vortex shedding and blade-wake interactions.

Consequently, two gaps in the literature emerge: i) the gap between BEM-based models and blade-resolved CFD in modelling hydrokinetic turbines in arrays with free water surfaces, and ii) the need for further development and validation of models with varying levels of fidelity to simulate HKTs under realistic conditions. Addressing these gaps could significantly influence the guidelines that are currently in use in the design of the Darrieus HKTs in arrays.

3 ORIGINAL CONTRIBUTION

3.1 List of Publications

During the PhD period, original contributions led to 10 publications, of which the following six represent the main findings of this thesis. While the other publications may indirectly contribute to the thesis objectives, they are not part of its core body.

- P1. Mohamed, O.S., Melani, P.F., Balduzzi, F., Ferrara, G., & Bianchini, A. (2023). An insight on the physical mechanisms responsible for power augmentation in a pair of counter-rotating Darrieus turbines. *Energy Conversion and Management*, 284, 116991. DOI: 10.1016/j.enconman.2023.116991
- P2. Mohamed, O.S., Melani, P.F., Soraperra, G., Brighenti, A., Balduzzi, F. & Bianchini, A. (2025). Selection criteria for rotor number and rotational direction in arrays of closely spaced Darrieus turbines. *ASME Journal of Fluids Engineering*, 147(3), DOI: 10.1115/1.4067007
- P3. Mohamed, O.S., Melani, P.F., Balduzzi, F., Ferrara, G., & Bianchini, A. (2022). An insight on the key factors influencing the accuracy of the actuator line method for use in vertical-axis turbines: Limitations and open challenges. *Energy Conversion and Management*, 270, 116249. DOI: 10.1016/j.enconman.2022.116249
- P4. Mohamed, O.S., Melani, P.F., Papi, F., Balduzzi, F., & Bianchini, A. (2023). Is the Actuator line method able to reproduce the interaction between closely-spaced Darrieus rotors? A critical assessment on wind and hydrokinetic turbines. *Energy Conversion and Management*, 293, 117473. DOI: 10.1016/j.enconman.2023.117473
- P5. Mohamed, O.S., Melani, P.F., Soraperra, G., Brighenti, A., Ferrara, G., Betti, V., Schippa, L., Guerrero, M., Balduzzi, F. & Bianchini, A. (2024). Three-dimensional CFD-ALM-VOF modelling of hydrokinetic turbines in realistic open-channel conditions. *Ocean Engineering*, 313, 119411, DOI: 10.1016/j.oceaneng.2024.119411
- P6. Mohamed, O.S., Melani, P.F., Soraperra, G., Brighenti, A., Ferrara, G., Betti, V., Balduzzi, F. & Bianchini, A. (2025). Development of a Computationally Efficient CFD Model for Hydrokinetic

vertical-axis turbines: A Critical Assessment. *Ocean Engineering*, 324, 120692, DOI: 10.1016/j.oceaneng.2025.120692

Other publications

- Ibrahim, A.A., Elbaz, A.M.R., Melani, P.F., Mohamed, O.S., Bianchini, A. (2022). Power augmentation of Darrieus wind turbine blades using trapped vortex cavity. *Journal of Wind Engineering and Industrial Aerodynamics*, 223, 104949, DOI: 10.1016/j.jweia.2022.104949
- Melani, P.F., Mohamed, O.S., Cioni, S., Balduzzi, F., and Bianchini, A. (2024). Tuning the actuator line method to properly modelling tip effects in finite-length blades. *Journal of Physics: Conference Series*, 2767, 052018, DOI: 10.1088/1742-6596/2767/5/052018
- Mohamed, O.S., Melani, P.F., Bangga, G., Aryan, N., Greco, L., and Bianchini, A. (2024). Accuracy assessment of Beddoes-Leishman and IAG dynamic stall models for wind turbine applications. *Journal of Physics: Conference Series*, 2767, 052053, DOI: 10.1088/1742-6596/2767/5/052053
- Melani, P.F., Mohamed, O.S., Cioni, S., Balduzzi, F., and Bianchini, A. (2024). An insight into the capability of the actuator line method to resolve tip vortices. *Wind Energy Science*, 9(3), DOI: 10.5194/wes-9-601-2024

3.2 Discussion

This section includes a discussion on the original contribution of this thesis, and it serves as the linkage between the main findings in the publications in Section 3.1. It starts with a discussion on the power augmentation mechanism in arrays of Darrieus turbines [P1, P2], then it delves into the main findings highlighted during the ALM tuning [P3, P4], and finally, it analyses the differences between different modelling approaches for Darrieus HKT arrays under realistic conditions [P5, P6].

3.2.1 Physics of Darrieus turbine in arrays

The power augmentation mechanism in closely spaced Darrieus turbines, identified as a key gap in the literature (Section 2.1), was thoroughly examined in [P1, P2]. These studies focused on analysing the relation between the instantaneous blade angle of attack (AoA), α in stand-alone and array configurations, revealing efficiency improvements based on the rotational direction relative to adjacent turbines. Using a two-bladed hydrokinetic turbine model, two-dimensional blade-resolved uRANS simulations provided valuable insights for optimizing turbine arrays.

The study in [P1] aimed to understand the power improvement mechanisms in closely spaced counter-rotating Darrieus turbine pairs, focusing on flow interactions within the mutual interaction region. It examined how these interactions influence the inflow local to the blade, a critical factor in turbine performance. Instantaneous AoA and blade forces were analysed over a full rotation to quantitatively characterise this mechanism. Building on this, [P2] extended the

analysis to co-rotating turbine pairs and three-turbine arrays, a relatively novel configuration that remains underexplored in the literature.

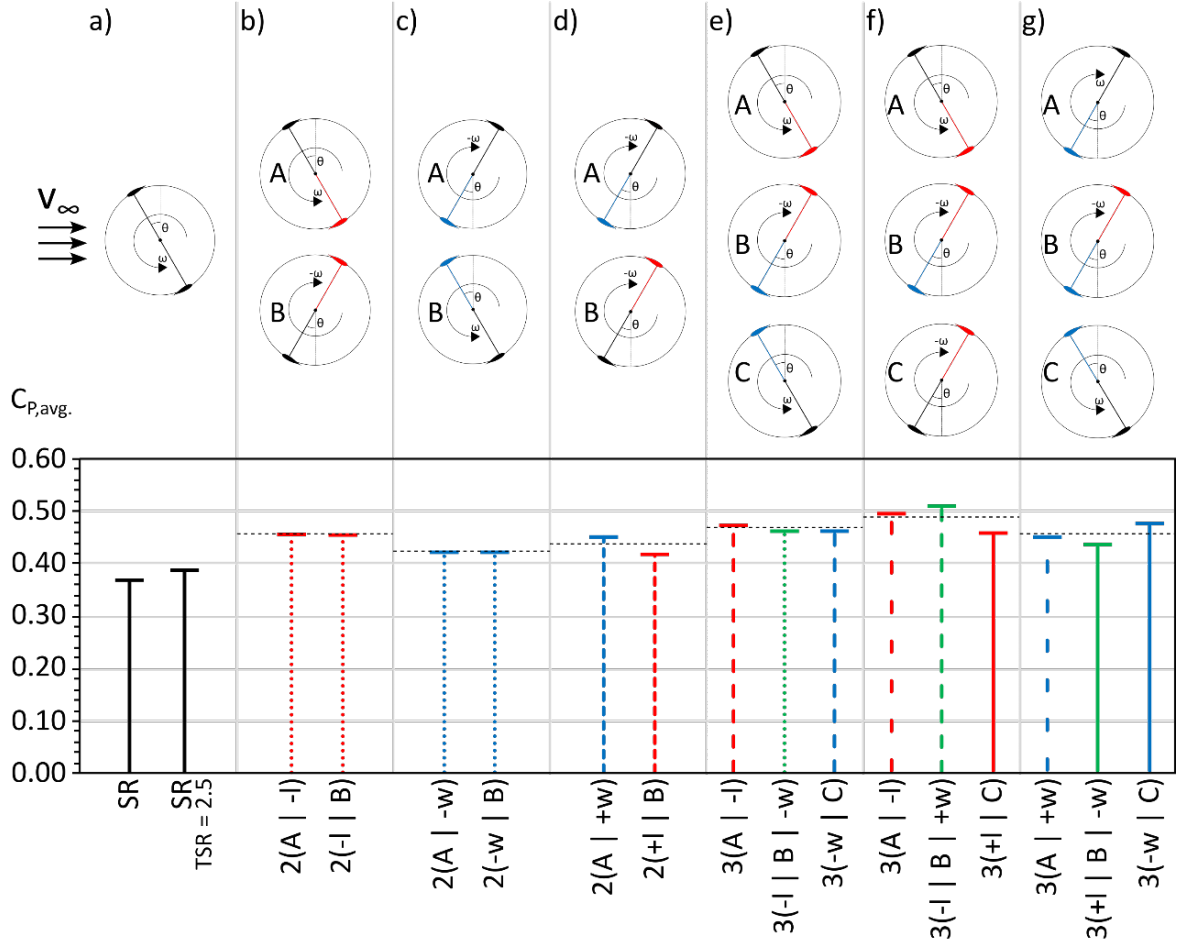


Figure 3.1: Mean power coefficient obtained for each turbine for different configurations of turbine arrays at $TSR = 3$ compared to that of the stand-alone turbine

Figure 3.1 compares the average power coefficient, $C_{p,avg.}$, for each turbine within different arrays, up to a three-turbine array, at a TSR of 3, which is identified as the optimum TSR for arrays of this turbine model, against the performance of a stand-alone turbine at both $TSR = 2.5$ (the optimum for an isolated turbine) and $TSR = 3$. To focus the analysis at the blade level, each turbine within the array was assigned an acronym based on four criteria. First, the number of turbines in the array is explicitly indicated at the beginning of the acronym. Second, the turbine under investigation is identified by its alphabetical order (e.g., A, B, or C). Third, the rotational direction relative to neighbouring turbines is denoted as either negative ("−") for counter-rotation or positive ("+") for co-rotation. Finally, the mode of mutual interaction is specified as either inward ("l") or windward ("w").

Findings from [P1] provide new insights into the mechanisms behind performance enhancement in closely spaced turbine arrays. While previous work by Alexander et al. [30] linked power augmentation to increased momentum entering the rotor domain, [P1] demonstrates that the azimuthal increase in momentum does not consistently coincide with higher efficiency positions, indicating that increased momentum alone does not explain power gains. Instead, the enhancement is primarily driven by aerodynamic changes, specifically the

reorientation of the relative velocity vector, which increases the angle of attack and lift generation on the blades—an effect also highlighted by Zanforlin et al. [29]. Figure 3.2 shows the instantaneous AoA and torque coefficients for turbines experiencing different modes of mutual interaction—leeward, windward, or both—compared to a stand-alone turbine. Two key observations emerge: first, torque increases directly with AoA, which is influenced by local flow changes due to turbine interactions; second, a consistent AoA increase pattern appears across all interaction modes, largely independent of the number of turbines, though the effect slightly strengthens with larger arrays.

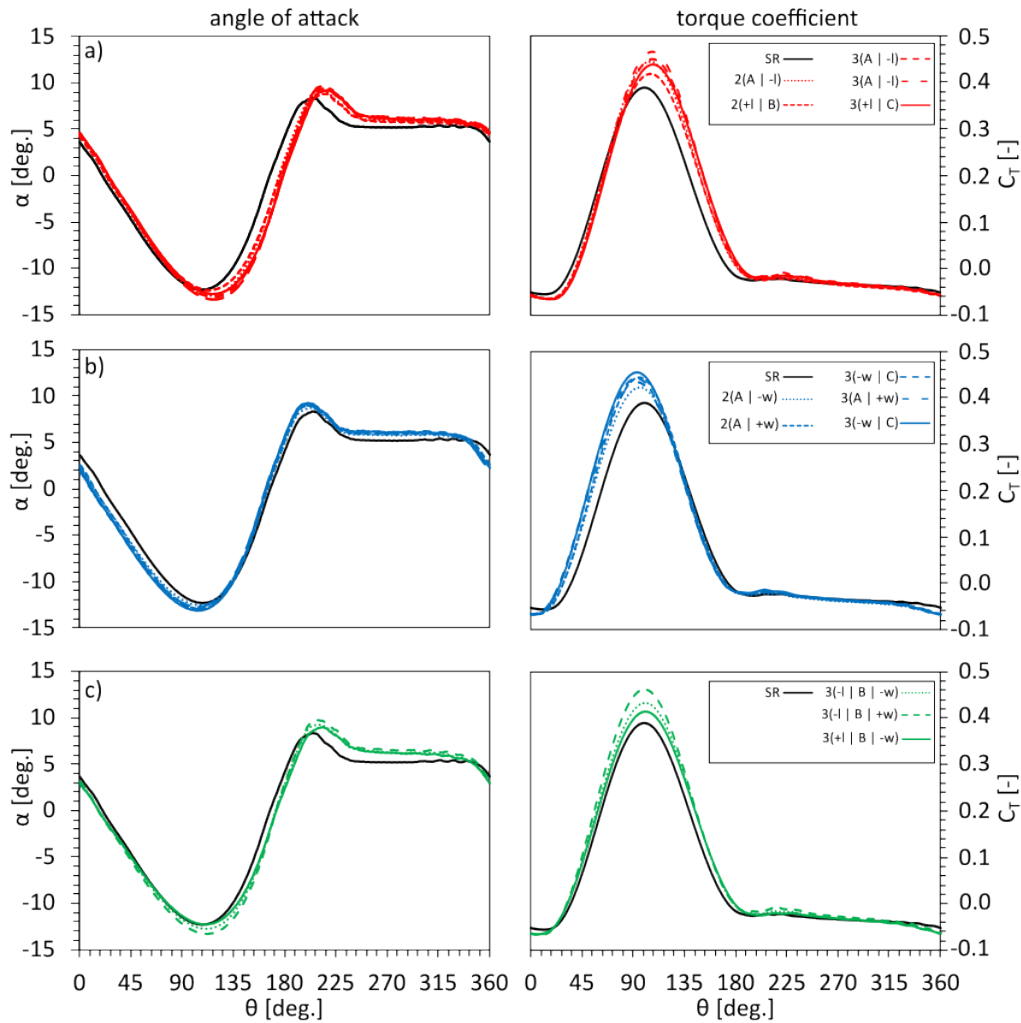


Figure 3.2: Instantaneous angle of attack α and torque coefficient C_T for different turbines within different arrays.

Velocity contours and streamlines are presented in Figure 3.3 for selected cases, illustrating how changes in the absolute inflow direction suppress the streamtube expansion effect. This suppression contributes to an increase in lift due to the increase in the AoA caused by the inflow straightening. In the inward configuration, the AoA increase occurs during the pitch-down phase, resulting in higher lift compared to an isolated rotor. In the outward configuration, the AoA increase takes place during the pitch-up phase, leading to a moderate lift improvement. These effects become more pronounced at higher rotational speeds, shifting the optimal power coefficient to higher TSRs. This increase in the optimal TSR is consistent with the findings in [26,27,29].

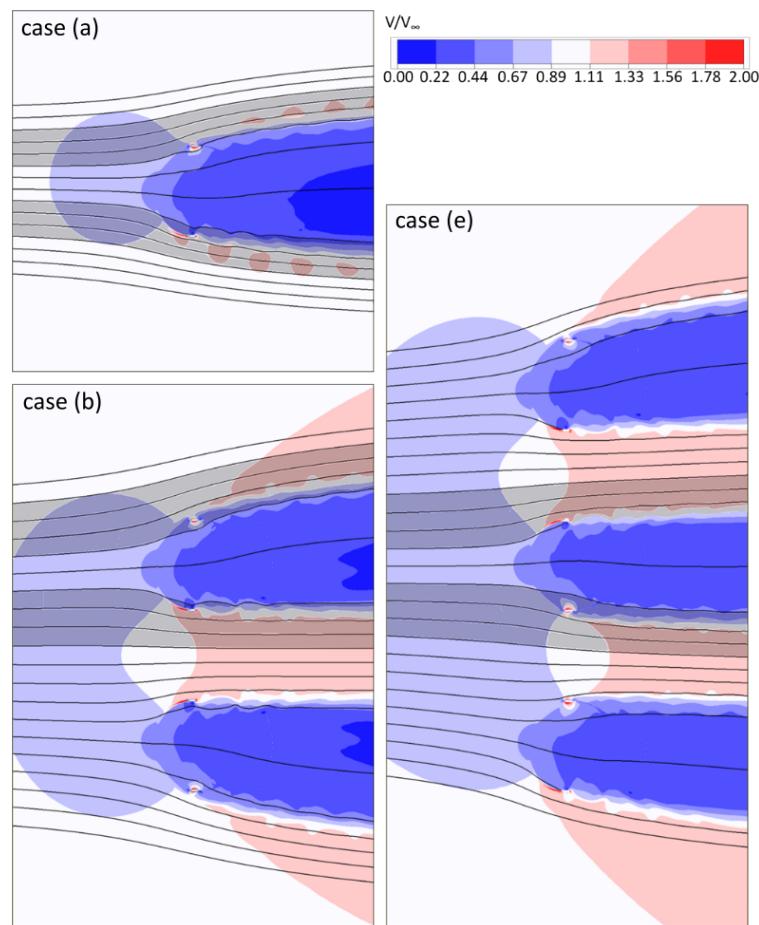


Figure 3.3: Velocity contours and streamlines for the stand-alone turbine cases (a) compared with cases (b) and (e) for arrays of two and three turbines, respectively.

The observed power augmentation mechanism follows a clear sequence: the presence of adjacent rotors induces a blockage effect, eliminating streamtube expansion and suppressing cross-stream velocity. This straightens the inflow, increases the AoA, and enhances lift and tangential forces, ultimately boosting power output. These effects are especially pronounced in the inward configuration, where most power generation occurs within the mutual interaction region. The mechanism of this effect, despite being discussed previously in [29,30], the results in [P1, P2] better articulates it by providing, for the first time, quantitative data on kinematic and force variations, in addition to the qualitative analysis. The increase in AoA is identified as the primary driver of power augmentation, offering valuable insights for future design optimisations, such as aerofoil selection, pitch control, and the use of flow deflectors [91], [31]. Additionally, these findings provide a foundation for validating and refining low-fidelity models for closely spaced Darrieus turbines [18,33].

The studies in [P3, P4] conclude that Darrieus turbines in a counter-rotating leeward configuration achieve the highest power augmentation, followed by co-rotating turbines with moderate efficiency improvements, and counter-rotating windward turbines, which show lower gains compared to stand-alone rotors. These findings align with previous research by Hara et al. [92] and Sahebzadeh et al. [23]. However, the studies acknowledge that three-dimensional flow effects likely reduce power augmentation due to faster wake recovery, as highlighted by Nazari et al. [93]. Experimental results from Müller et al. [20] further support this, demonstrating

greater wake diffusion in co-rotating turbine pairs. To enhance efficiency and wake recovery in multi-turbine arrays, the study recommends favouring leeward interactions and co-rotating configurations.

3.2.2 ALM tuning in open-field conditions

The ALM is one of the main tools employed in this thesis. An extensive tuning process for the ALM was undertaken, with the results published in [P3, P4]. Initially, a detailed assessment was performed to evaluate the influence of various factors on the accuracy of the method in simulating the performance of a stand-alone Darrieus turbine. This was followed by a comprehensive validation campaign for the two-dimensional implementation of the ALM, ensuring its ability to accurately reproduce the power augmentation mechanisms observed in turbine arrays. To enhance the reliability of the evaluated parameters and support broader applicability of the results, two distinct Darrieus rotors were analysed: a wind turbine and a hydrokinetic turbine. This selection was based not only on the extensive and high-quality benchmark data available, but also on the potential to extend the findings to a wide range of turbine architectures, solidities, and aerofoil profiles.

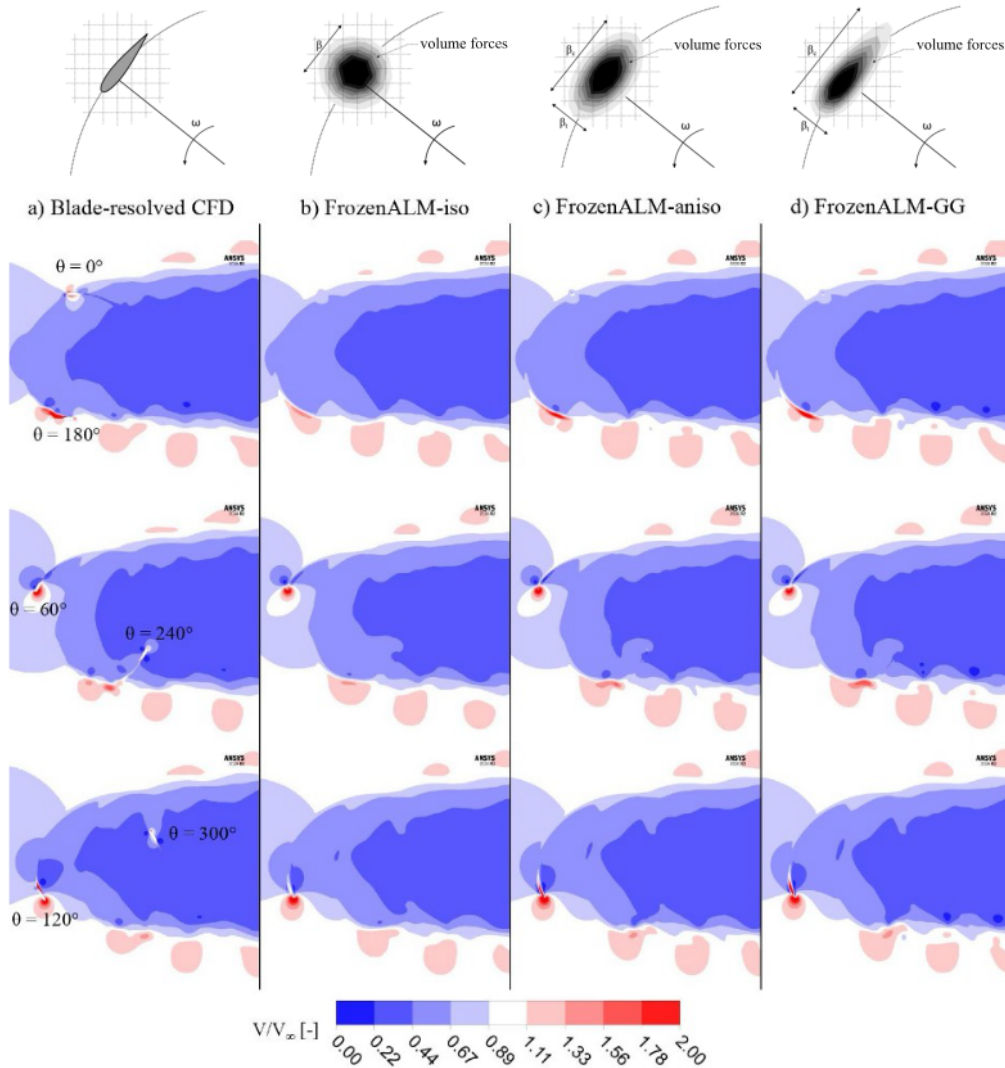


Figure 3.4: dimensionless velocity contours showing the Frozen ALM concept used to study different Regularization kernel shapes.

Two versions of the Actuator Line Model (ALM) were employed during the tuning process: the standard ALM (see Section 2.2.4) and the Frozen ALM concept. The latter involves inputting forces directly extracted from high-fidelity blade-resolved CFD simulations. In this process, a blade-resolved uRANS simulation of the case under study is performed, where the time history of forces on each blade is computed by resolving the boundary layer along the blade surface. These force time series are then used as input for an ALM simulation under identical operating conditions. This method enables the isolation of uncertainties related to the quality of polar data and the modelling of dynamic stall.

Figure 3.4 illustrates the use of the Frozen ALM to evaluate different functions for spreading the forces, namely isotropic Gaussian, anisotropic Gaussian, and anisotropic Gaussian Gumbel. The results of this analysis reveal that using different kernel shapes to distribute the forces does not significantly influence either the flow field (as depicted in Figure 3.4) or the performance predictions. The latter observation has been debated in various sources in the literature for HATs [94–96].

Building on the work of Melani et al. [58], the findings in [P3] offer a comprehensive evaluation of the factors influencing the accuracy of the ALM for stand-alone turbines. The study focuses on input polar data, dynamic stall modelling, and the Gaussian function used to distribute the forces. While the kernel shape was observed to have minimal impact on the ability of the ALM to predict blade loads and wake flow fields, the results highlight a strong dependency on the dynamic stall model. The accuracy of predictions varies significantly with the operating TSR, with greater discrepancies between ALM and blade-resolved CFD simulations occurring at lower TSRs, where dynamic stall effects are more pronounced. Additionally, grid resolution was identified as a critical factor for accurately reproducing the flow field, with optimal grid sizes determined for different kernel functions.

Figure 3.5 presents a bubble chart summarising the factors influencing ALM accuracy, based on assessments conducted on two different turbines. These findings and observations enhance the reliability of the ALM, particularly for complex applications, including not only hydrokinetic systems but also other challenging scenarios such as floating offshore wind energy and urban wind energy.

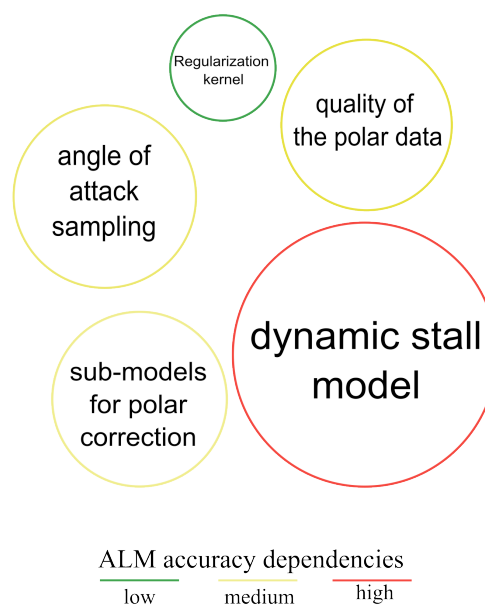


Figure 3.5: Bubble chart showing ALM accuracy dependencies.

The study in [P4] builds upon the work in [P3] and provides a comprehensive validation of the two-dimensional ALM for closely spaced twin Darrieus turbines. It demonstrates the capability of the ALM model to capture the complex flow interactions observed in turbine pairs, which are typically analyzed through blade-resolved CFD. This step was crucial, given the lack of similar analyses in the literature, and was essential for confirming that the power augmentation mechanism observed in Darrieus turbine arrays in [P1] and [P2] via blade-resolved modeling can indeed be replicated using the ALM. This finding also holds significant industrial relevance, positioning the ALM as a computationally efficient alternative that could serve as the standard for applications such as HKTs deployed in arrays. The validation showed that the ALM accurately reproduces local flow kinematics, critical for closely spaced turbine configurations, and effectively estimates instantaneous turbine loads with reasonable accuracy.

Figure 3.6 compares the performance of a stand-alone turbine with that of a turbine in an array of two turbines, showing the instantaneous torque coefficient estimated by both blade-resolved CFD and the ALM for an HKT turbine pair and a VAWT pair. As shown, the ALM accurately captures the variation in the torque coefficient for both turbine models, particularly during the upwind half of the rotation, and it is clearly capable of capturing the surplus power due to the turbines being installed in close proximity. However, during the downwind half, the ALM slightly overestimates the torque coefficient, a discrepancy attributed to its limited ability to fully capture blade-wake interactions.

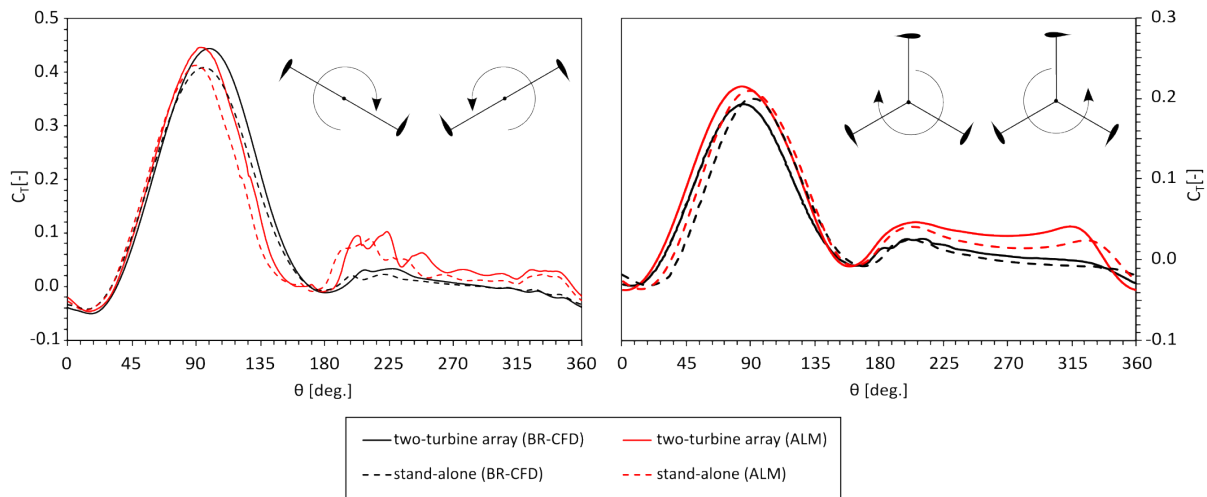


Figure 3.6: Comparison between the estimations of ALM vs blade-resolved CFD for the load in terms of moment coefficient for the HKT pair at TSR = 3 (left) and the VAWT pair at TSR 2.7 (right).

The wake behaviour behind the Darrieus HKT pair, as predicted by the ALM, Frozen ALM, and blade-resolved CFD, was analysed using Proper Orthogonal Decomposition (POD), an effective method for assessing the ability of the ALM to reproduce the wake flow field with respect to the blade-resolved CFD. Figure 3.7 presents the POD modes ranked by modal energy for the fields obtained from blade-resolved CFD, Frozen ALM, and ALM for the HKT pair at TSR = 3. As shown, the Frozen ALM closely matches the blade-resolved CFD in both the energy content of each mode and the modes themselves. This indicates that the discrepancies observed in the ALM results are related to the computation of forces, specifically either the quality of the polar data or deficiencies in predicting unsteady forces.

In short, the extensive tuning and validation efforts undertaken for the ALM in this thesis have demonstrated its significant potential as a reliable and efficient tool for simulating Darrieus

rotors, both individually and in array configurations. The use of the Frozen ALM approach has provided a deeper understanding of the model accuracy, particularly in separating the effects of aerodynamic data quality and unsteady forces modelling. The ALM was found capable of providing accurate predictions of rotor performance, local flow kinematics, and wake behaviour, with computational efficiency far surpassing that of conventional blade-resolved CFD. While the accuracy of the ALM is influenced by factors such as grid resolution and dynamic stall model, its ability to capture complex flow interactions with minimal computational resources makes it an attractive option for large-scale simulations. This is particularly relevant for advanced applications, which are not limited to hydrokinetic systems, despite the core focus of this thesis, but also extend to floating offshore turbines and urban wind energy systems. Future improvements in dynamic stall modelling and further validation with three-dimensional simulations and experimental data could enhance the ALM applicability and precision in real-world scenarios.

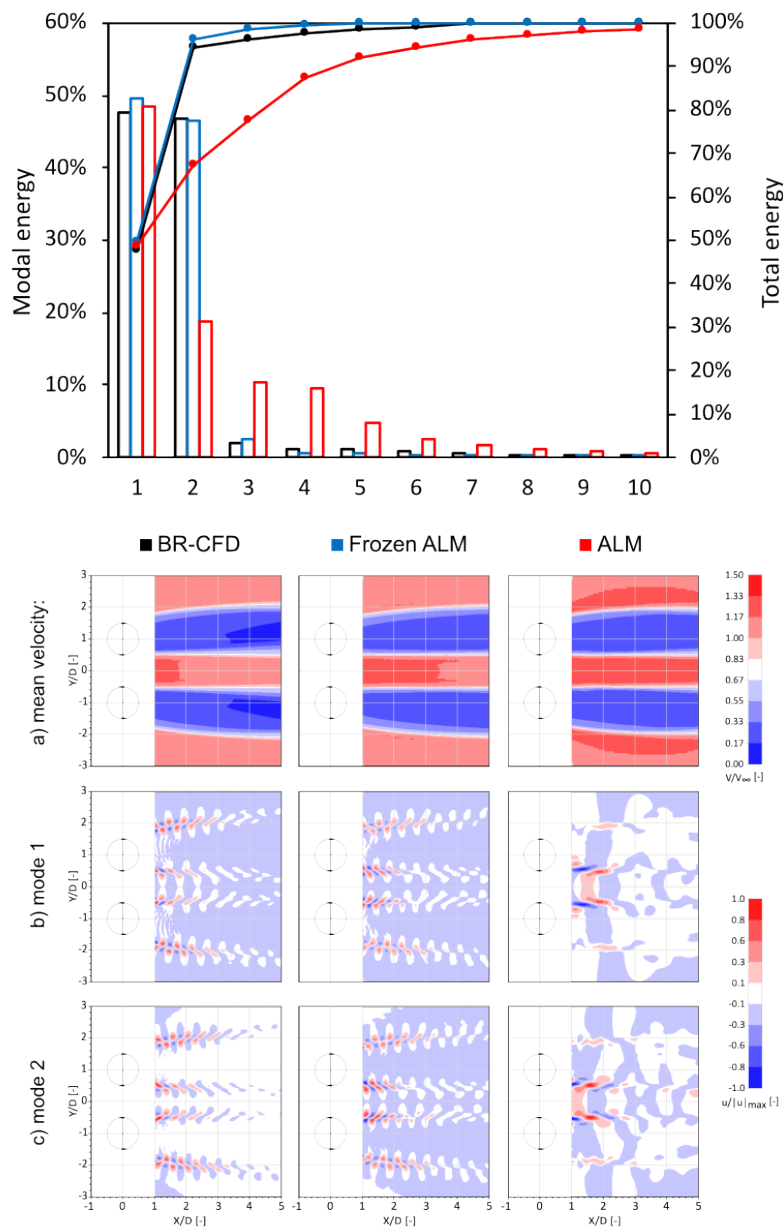


Figure 3.7: Results of the POD wake assessment: BL-CFD vs ALM vs Frozen ALM

3.2.3 Modelling and validation of Darrieus HKTs under realistic conditions

As detailed in Section 2.3, modelling arrays of HKTs while considering the hydraulic complexities is either computationally expensive, making simulations of arrays unfeasible, or reliant on low-fidelity models such as the DMST, which may be unreliable due to the complexity of Darrieus turbine arrays and their interaction with the channel resource. In P[5, 6], two novel models based on ALM and AC, respectively, were developed to simulate Darrieus HKT arrays in realistic open-channel conditions using the VOF method. Both models were validated against experimental measurements and underwent a detailed code-to-code comparison. These models fill a gap in the literature by providing accurate results close to experimental data, particularly the ALM, while also offering a significant reduction in computational time, especially with the AC model.

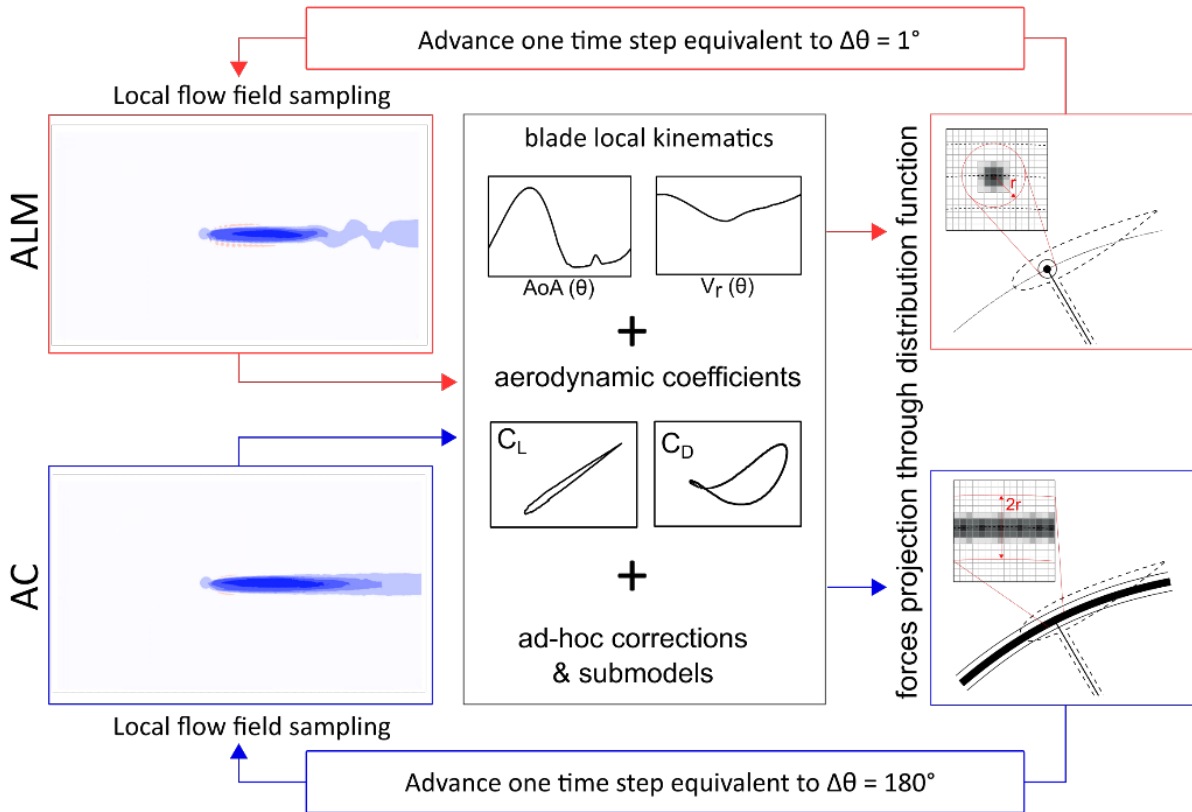


Figure 3.8: Schematic representation of the differences and similarities between the ALM and AC turbine models used in [P5] and [P6], respectively.

The experimental measurements used to validate the latter models were obtained by the industrial partner, HE-PowerGreen, for the Biffis Canal [97], located in Trento, Italy. These measurements included the performance of a turbine within an array of four Darrieus turbines as illustrated in Figure 3.9, the water level rise upstream of the turbines, and Acoustic Doppler Current Profiler (ADCP) measurements of the flow field upstream and downstream of the array.

Following the fine-tuning and sensitivity analyses carried out for the ALM model, [P5] uses the fully developed version of the code to model an array of Darrieus turbines within a channel, coupled with the VOF approach to model the free water surface variation. On the other hand, [P6] presents the AC-based code, developed as a simplification of the ALM and coupled with the VOF. In this simplified model, only average forces are computed and projected into the CFD

domain, making the turbine model independent of time-step size and allowing the use of larger step sizes, significantly reducing computational time. The key difference between the ALM-VOF and AC-VOF models lies in how the turbine is represented, although the same ad-hoc corrections and sub-models were used. Figure 3.8 compares the workflows of both the AC and ALM models.

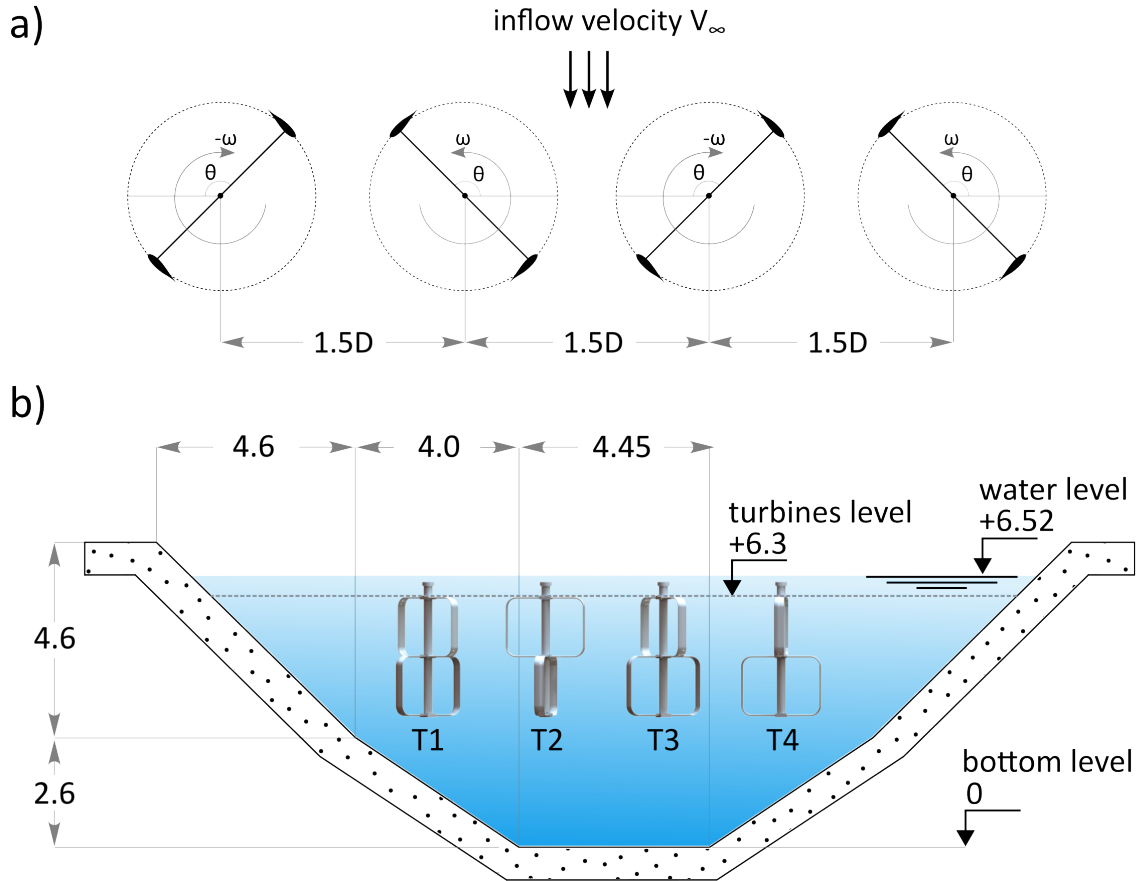


Figure 3.9: (a) Tested array configuration for the installed turbines and (b) schematic drawing for the Biffi canal section and the installed Darrieus turbine array.

The study in [P5] introduced the CFD-ALM-VOF model, which offers a good balance between accuracy and computational efficiency. The model was successfully validated against experimental measurements in terms of turbine performance, water levels, and flow velocities, showing that closely spaced turbines could generate up to 25% more power than standalone turbines due to blockage effects. The model showed improved performance predictions at higher TSRs by accounting for water surface variations, though its accuracy decreased at low TSRs due to unsteady flow effects. It found good agreement with experimental velocity measurements, particularly in the near wake, although discrepancies at the turbine tip were noted, indicating the need for better modeling of turbine struts. The research also highlighted the importance of considering inlet velocities when simulating HKTs, as blockage effects could overestimate power coefficients.

In [P6], a quasi-steady CFD-AC-VOF model was presented. The model was tuned for key parameters such as flow field sampling, kernel size, and actuator point resolution. A code-to-code comparison with the ALM assessed the capabilities of the AC model in estimating channel blockage and power augmentation in arrays compared to standalone turbines. Key findings include strong agreement between the AC and ALM models for standalone turbines, with

minimal turbine-to-channel interactions. In array configurations, the ALM model provided more accurate results, closely matching experimental data. The AC model slightly overestimated the channel blockage effects and turbine-to-turbine interactions, underestimating the power gain from closely spaced turbines. Despite these discrepancies, the AC model still provided reasonable estimations of azimuthally dependent quantities, making it suitable for rapid force estimations.

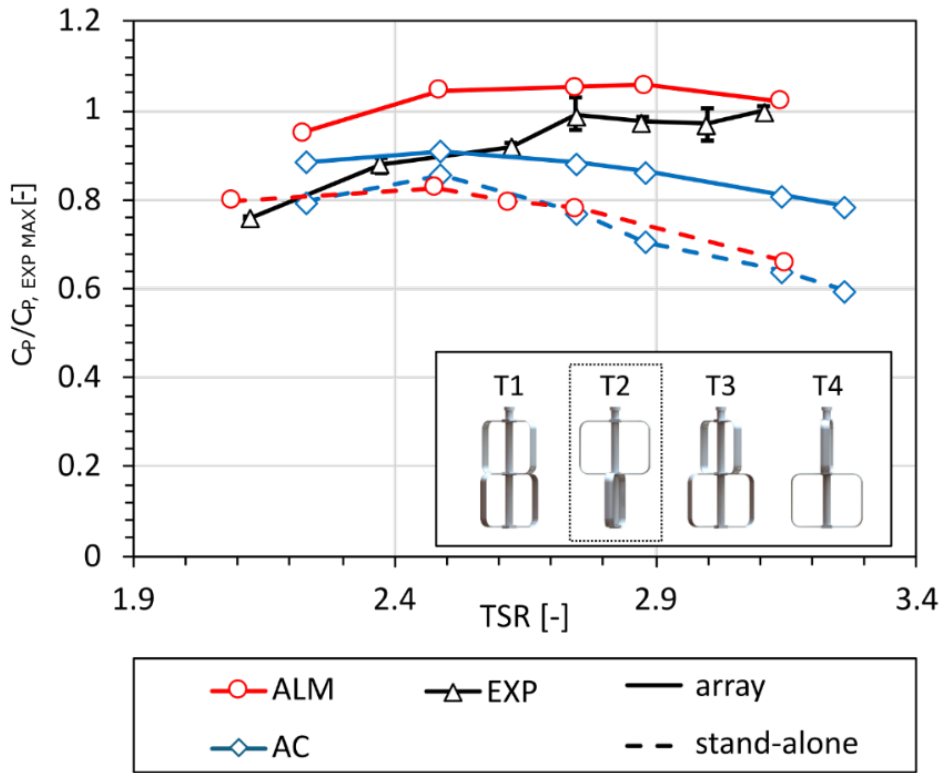


Figure 3.10: Comparison of the normalised power coefficient values between the ALM and AC models for the stand-alone turbine and turbine T2 in the array, against experimental measurements for turbine T2.

Figure 3.10 shows the average power coefficient, C_p of the stand-alone turbine and turbine T2 in the array, estimated by the AC and ALM models, compared with experimental measurements for T2. All results are normalised by the maximum experimental C_p for industrial confidentiality concerns of the industrial partner, with no impact on the comparison. For the stand-alone turbine, both models agree closely, with a peak C_p at $TSR = 2.5$ and a maximum relative difference of 3.4%. However, in the array performance, the AC model underestimates C_p at higher TSRs, reaching a 20.6% error at $TSR = 3.14$, while the ALM model aligns better with experimental data, showing a maximum error of 6%. At lower TSRs, the AC model performs better, but struggles with capturing the trend of C_p variation, while the ALM model more accurately reflects turbine-to-turbine interactions. Both models degrade in accuracy at lower TSRs due to unsteady flow effects, which are not fully captured by the Berg dynamic stall model, as discussed earlier in [P1].

Upon careful analysis, it was found that the increase in the AoA associated with power augmentation in Darrieus turbine arrays is not well captured by the AC model. This is attributed to the steady-state nature of the model, which cannot fully account for the increase in AoA in the mutual flow interaction areas between adjacent turbines. Additionally, the hydraulic effects due

to interactions between the channel and the array are overestimated in the AC model, resulting in more blockage than in the ALM model. This leads to a higher upstream water rise in the AC simulation, while the flow velocity is reduced compared to the ALM.

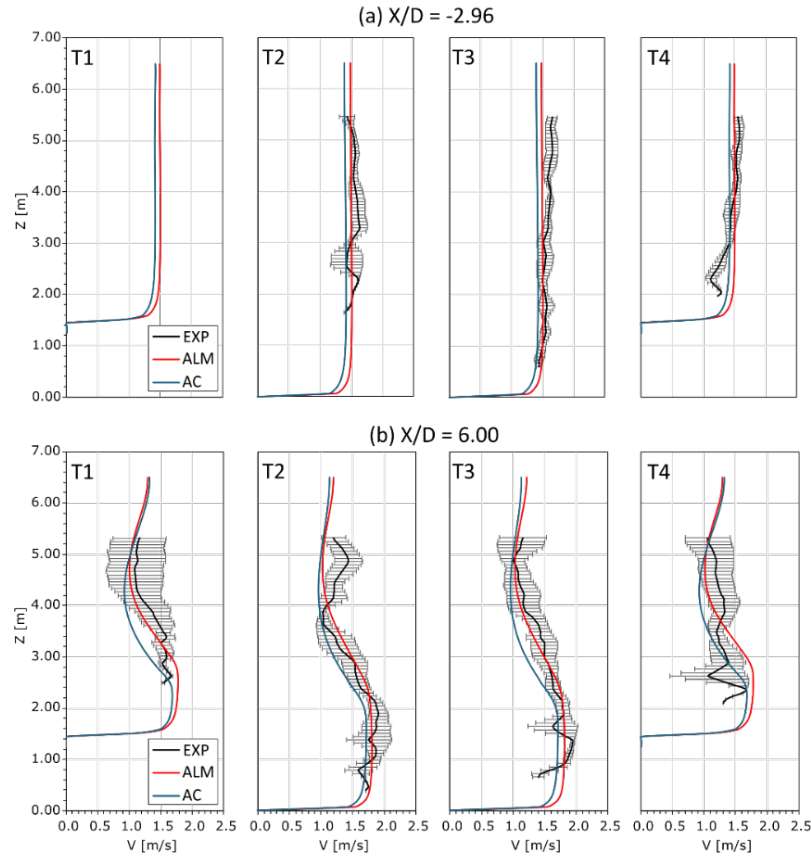


Figure 3.11: Comparison of longitudinal velocity profiles from the AC and ALM models with experimental data upstream and downstream of the turbine array at $TSR = 2.75$.

Figure 3.11-a presents the time-averaged vertical velocity profiles measured upstream of each turbine at $X/D = -2.96$, along with results from the AC and ALM simulations. Both models predict nearly uniform velocity, while experimental measurements show slight variations. The AC model predicts a slightly greater velocity deficit upstream due to a higher rise, but overall, the numerical and experimental profiles show good agreement, with minor discrepancies near the walls, likely due to boundary conditions and measurement uncertainties. Figure 3.11-b shows the downstream velocity deficit at $X/D = 6$, where a clear deficit is seen in the wake, along with areas of acceleration due to flow interactions and blockage effects. The AC model predicts a higher velocity deficit than the ALM, especially downstream of the lower rotor. While the profiles downstream of the upper rotor show good agreement across all methods, greater divergence is observed downstream of the lower rotor, with the ALM results aligning more closely with experimental data.

The discrepancies between the ALM and AC models, particularly in array simulations, are highlighted in the dimensionless velocity contours in Figure 3.12. For the stand-alone turbine, the wake flow fields of both models are nearly identical, reflecting the low blockage effect of a single turbine. In the array, however, the AC model shows lower flow acceleration between turbines, indicating underestimation of turbine-to-turbine interactions and power coefficient.

Upstream, the AC model predicts a lower inflow velocity compared to the ALM, suggesting an overestimation of blockage effects due to array-channel interactions.

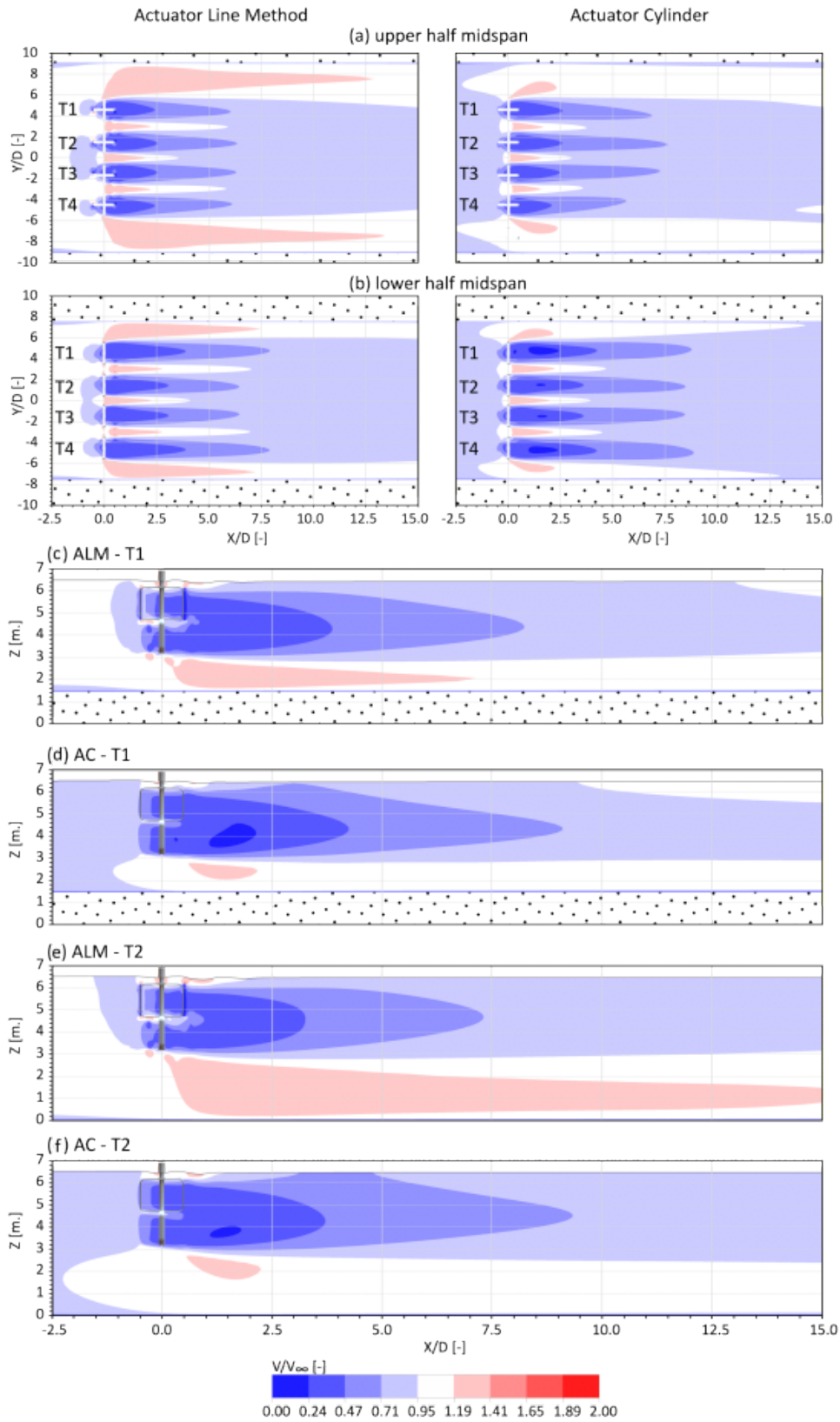


Figure 3.12: Dimensionless velocity contours for a stand-alone turbine and a 4-turbine array comparing the Actuator Line Method to the Actuator Cylinder at a Tip Speed Ratio of 2.75.

The computational times of the ALM and AC models demonstrate distinct scaling behaviours. The AC model, while requiring more iterations to achieve convergence, completes simulations significantly faster overall. Its computation time scales directly with the number of turbines, with a 4-turbine array taking approximately four times longer than a stand-alone turbine. In contrast, the ALM exhibits a less linear scaling pattern, with the array requiring about twice the computation time of a stand-alone turbine. Table 3.1 provides a detailed comparison of these computational costs.

Table 3.1: Comparison in terms of computational cost between AC and ALM.

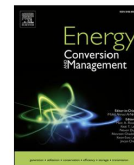
Value	AC		ALM	
	Stand-alone	array	Stand-alone	array
Total simulation time [h/core]	423	1,824	6,132	10,578
Simulated flow time [s]	≈ 70		≈ 30	

When compared to prior studies, such as those by Gauvin-Tremblay and Dumas [79] and Yosry et al. [82], the presented models offer a more computationally efficient alternative. The earlier works relied on blade-resolved CFD to study turbine performance and its interaction with the channel, which, although scientifically valuable, are computationally prohibitive, especially for turbine arrays. By employing medium-fidelity tools, the ALM and AC models achieve similar analyses with significantly reduced computational demands while maintaining reasonable accuracy. These advancements highlight the practicality of the presented models for studying hydrokinetic energy systems, especially within the industrial sector.



Contents lists available at ScienceDirect

Energy Conversion and Management

journal homepage: www.elsevier.com/locate/enconman

An insight on the physical mechanisms responsible of power augmentation in a pair of counter-rotating Darrieus turbines

Omar S. Mohamed, Pier Francesco Melani, Francesco Balduzzi, Giovanni Ferrara, Alessandro Bianchini*

Department of Industrial Engineering, Università degli Studi di Firenze, Via di Santa Marta 3, 50139 Firenze, Italy

ARTICLE INFO

Keywords:

Darrieus
Vertical Axis Wind Turbines (VAWT)
Hydrokinetic energy
Angle of attack
CFD

ABSTRACT

In recent years, Darrieus turbines have received increasing attention by both the industrial and the academic sector due to their advantages for both wind and hydrokinetic applications. Experimental and numerical investigations showed that the interaction between closely spaced Darrieus rotors can lead to a significant increase in their efficiency. To date, however, the physics underlying this phenomenon has been only argued or qualitatively discussed, since no robust method to determine the actual angle of attack was available.

This study provides a novel insight into the physical mechanisms that lead to performance enhancement in twin Darrieus rotors. A pair of counter-rotating, two-blade Darrieus hydrokinetic turbines are investigated while spinning in both the revolution senses, namely, *inward* and *outward*, by means of unsteady Computational Fluid Dynamics. After a preliminary analysis on the effect of rotor spacing, an improved flow sampling method is used to analyze the instantaneous flow kinematics past the blades in combination with computed blade loads, thus allowing the calculation of lift and drag forces. Results show that the optimum center-to-center distance for the turbines under consideration is 2D. Also, it is shown that the optimum operating point of the twin rotors tends to shift toward higher Tip-Speed Ratios (TSRs). Compared to the stand-alone turbine, the efficiency improves by 16.1% and 8.7% for the *inward* and *outward* twin rotors setups, respectively. A detailed study of the local flow field shows a complete suppression of the streamtube expansion within the area of mutual interactions between the adjacent rotors. This allows more momentum flux to enter the rotor area compared to that of the isolated rotor. More interestingly, it is observed that the change in the inflow velocity direction leads to an increase in the angle of attack for both the *inward* and *outward* setups, causing an increase in the generated lift, which is found as the main reason for the efficiency improvement.

1. Introduction

Research interest in Vertical Axis Wind Turbines (VAWTs) has been increasing significantly during the last couple of decades, targeting different potential applications, from small-scale urban power generation [1] up to large-scale floating offshore installations [2]. These machines are known for the independence of their performance on the wind direction and the low-noise operation. Above all, they come with the unique benefit of having the torque transmitted through a vertical shaft, thus providing the possibility of positioning the powertrain not at height like in their horizontal-axis counterparts. The latter advantage makes the Darrieus turbine economically attractive for some applications such as Floating Offshore Wind Turbines (FOWTs) [3], but also for Hydrokinetic Turbines (HKTs), since in this case the generator can be placed

out of the water, with huge benefits in terms of installation, reliability and cost. However, Darrieus turbines are still underperforming in terms of efficiency if compared to horizontal-axis ones [4].

Studies have found that the aero/hydrodynamic efficiency of Darrieus turbines can be improved if two or more rotors are deployed in close proximity, resulting in an increased power generation compared to a standalone turbine [5,6]. This characteristic not only minimizes the efficiency gap between Darrieus and horizontal-axis turbines, but also increases the power density (i.e., power output per occupied land area) of Darrieus turbines. In fact, this concept has attracted the interest of the industrial sector recently in different applications, such as Nenuphar's Twinfloat concept [7], which involves two floating Darrieus wind turbines sharing a common platform, and HE-Powergreen's array [8], which is composed of four hydrokinetic Darrieus turbines sharing the

* Corresponding author.

E-mail address: alessandro.bianchini@unifi.it (A. Bianchini).<https://doi.org/10.1016/j.enconman.2023.116991>

Received 16 February 2023; Received in revised form 29 March 2023; Accepted 30 March 2023

Available online 6 April 2023

0196-8904/© 2023 Elsevier Ltd. All rights reserved.

same supporting structure on a channel. However, a significant drawback of this compact setup is that the performance becomes more sensitive to the inflow direction, depriving the machine of its omnidirectional advantage. In the case of hydrokinetic turbines investigated in this study, characterized by the unidirectional nature of water streams, closely spaced Darrieus turbines could be an optimal solution for power generation.

Dabiri [5] has conducted a campaign of field tests to demonstrate that counter-rotating closely spaced Darrieus turbines produce more power compared to the stand-alone turbine. Vergaerde et al. [6] performed wind tunnel tests for a pair of VAWTs. They concluded an optimum center-to-center spacing of 1.2 and 1.3 rotor diameters, with a relative power increase of 16% compared to an isolated rotor. Li et al. [9] carried out an experimental investigation on the performance of twin Darrieus rotors in an urban environment; they in fact observed an improvement in the performance if the turbines are placed within a closely spaced distance. Müller et al. [10] experimentally analyzed the wake behind twin Darrieus hydrokinetic turbines. They concluded that the slower wake recovery was in the case where the blades are counter-rotating in the inflow direction, while pointing out that the faster wake recovery was in the case of counter-rotating against the inflow direction.

Besides experimental studies, several numerical investigations focused on investigating the optimal spacing and angle that should be kept between two or more Darrieus turbines. Mohamed et al. [11] concluded that closely spaced rotors are more efficient if they are deployed in-line with zero oblique angles. Zheng et al. [12] confirmed the efficient performance of the in-line setup and recommended a spacing distance between 1.5D and 1.6D to maximize the overall cluster efficiency. Sahebzadeh et al. [13] on the other hand studied the co-rotating configuration of a Darrieus turbine pair at different layout arrangements. They found that the co-rotating configuration can reach a maximum overall efficiency almost equal to that of the single rotor. Zhao et al. [14] concluded that adding a flow deflector between counter-rotating turbines could enhance the overall power coefficient by up to 23%. However, it makes the performance even more sensitive to the inflow direction.

Other numerical studies were conducted to understand the physical power augmentation mechanism of the twin Darrieus rotors. Through 2D RANS simulations, Zanforlin et al. [15] speculated that there are two reasons for power enhancement, namely the favorable deviation in absolute wind velocity, and the wake contraction in the downwind path that leads to an increase in the momentum flux available. Alexander et al. [16] extended the latter analysis by observing that the volumetric flow that enters the turbine increases when they are deployed closely due to the blockage effect occurring between the adjacent rotors. They also concluded that the majority of power coefficient enhancement occurs during the mutual interaction zone where the blades are closer to each other, whereas, during other azimuthal positions, the performance might be decreased.

In their study, Guilbot et al. [17] analyzed the performance of twin Darrieus rotors with two blades each. Their investigation focused on the impact of rotor distance and rotation orientation on overall performance. Meanwhile, Peng et al. [18] employed the Taguchi method to examine the influence of airfoil profile, pitch angle, solidity ratio, rotational direction, and spacing between the closely-spaced Darrieus turbines on the optimum Tip Speed Ratio (TSR). Their findings revealed that the solidity ratio has the most significant effect on changing the optimal TSR, which aligns with previous research on single rotors [19]. However, they also noted that the optimum TSR for twin rotors differed from that of a single rotor and tended to increase. Recently, Cacciali et al. [8] proposed an optimization tool to maximize the power output of an in-line array of closely-spaced two-bladed hydrokinetic Darrieus turbines using a Double Multiple Stream Tube (DMST) model incorporated with a confined environment flow field analysis to account for the turbine's fluid dynamic losses, turbulent mixing loss, and channel friction losses, allowing the estimation of backwater propagation.

However, beyond the cited works, to date there is a lack of real quantitative analyses regarding the kinematic behavior of turbines within the mutual interaction region. Most of the works in this area focus on optimizing the overall performance of the turbines, while ignoring the mechanism that leads to such enhancements. In spite of their completeness in terms of flow field analysis, none of previous works analyzed how the interaction between the two turbines actually affects the local inflow on the blades. Indeed, to further develop the concept of closely spaced Darrieus turbines, sufficient knowledge of the variation of the local flow kinematics is needed, which could be beneficial during the rotor design process, e.g., airfoil selection, pitch control, etc. In addition, this information can also be used for the validation and tuning of low- and medium-fidelity tools for the simulation of Darrieus turbines.

This limitation can be mainly attributed to a lack of consensus about the correct procedure to sample the angle of attack (AoA) from a Computational Fluid Dynamics (CFD) simulation. Among the most credited ones, one could indicate the *3-Points* [20], *Trajectory* [21], and *LineAverage (LineAvg)* [22]. Nevertheless, these methods have been originally developed and validated for Horizontal Axis Wind Turbines (HAWTs), with only few examples featured the use of these methods to sample the AoA for VAWTs. Among these, one could cite that of Elsakka et al. [23], where the AoA was sampled on two points upstream of the airfoil. However, the validation for this method was conducted on a static airfoil, which does not encompass the complex phenomena associated with functioning of a Darrieus rotor. Previous works by the authors in Ref. [24,25] compared different methods for sampling the AoA at different operating conditions of the Darrieus turbine, and it has been concluded that the *LineAvg* technique is the most effective among the AoA sampling methods, in terms of data smoothness and reliability. According to these studies, the *Trajectory* shows significant discontinuities near zero AoA, likely because it is an iterative technique and suffers from convergence issues. On the other hand, the *3-Points* or other similar variations (*2-Points* in Ref. [23]) underestimates blade AoA, particularly at low TSRs where flow separation vortices can overlap with one of the sampling points, creating disturbances in the AoA samples. This results in unphysical trends in the computed aerodynamic coefficients.

1.1. Objectives and outline of the study

This paper aims at a more profound physical understanding of the power improvement mechanisms in closely spaced Darrieus rotors, emphasizing the rotors' mutual interactions. This has been achieved by sampling the instantaneous flow velocities near the blade, which are then used to evaluate the amount of momentum flux that enters the rotor domain, quantified via the induction factor (a). AoA data, combined with computed blade forces, were then used to obtain the lift and drag forces on the turbine blade during a full revolution. The element of novelty in this paper is then the quantitative evaluation of how the interaction between two closely spaced Darrieus rotors affects the local blade inflow.

A couple of Darrieus rotors was selected as a study case and analyzed for different spatial arrangements and in the two possible orientations of the counter-rotating configuration, namely, *inward* and *outward*. Different spacings between the rotors were investigated, using only the inline arrangement. Results were then compared with the performance of the stand-alone rotor in terms of blade kinematics, namely the instantaneous angle of attack and torque on a single blade, and the total volumetric flux through the rotor area. Finally, the lift and drag characteristics were evaluated.

The paper is structured as follows. In Section 2, a description of the test case is provided by reporting the geometrical and operational characteristics of the turbine, together with the performance parameters used in the current analysis. In Section 3, the numerical methodology is discussed, describing the geometry and mesh properties as well as the chosen simulation setup. After that, the local flow sampling strategy (*LineAvg*) and its settings are described. Section 4 shows a selection of

main results into two parts. In the first one, the average and instantaneous power outputs for the tested cases are shown, comparing the performance of the counter-rotating turbines with that of the stand-alone one. In the second part, the kinematic characteristics are presented in the form of induction factor, AoA, and chord-based Reynolds number Re_c , followed by an analysis of the lift generation for a single cycle. Finally, conclusions and future work are discussed in Section 5.

2. Test case

2.1. Turbine model

The selected test case is a small-size, two-blade H-Darrieus hydrokinetic turbine designed with an industrial partner by some of the authors in a previous work [26]. This turbine has also been used as a case study for a new active pitch control concept in [27]. In addition, it was one of the turbines used to develop the medium-fidelity simulation tool, the Actuator Line Method (ALM) for Darrieus turbines in [28,29].

An illustration of the turbine geometry is shown in Fig. 1a. It is important to note that only two-dimensional geometry was considered in the present study. Turbine blades are based on a DU-06-W-200 airfoil with a chord length of 0.25 m. Turbine diameter is 2 m. Other geometrical characteristics are reported in Table 1. The optimum operating condition of the isolated rotor corresponding to a Tip-Speed Ratio (TSR) of 2.5 and a power coefficient of approximately 0.38 [26], as shown in Fig. 1b. More details on the operational characteristics are mentioned in Table 2. The twin-rotor configuration was simulated in the counter-rotating setup, taking both rotational orientations into account, i.e., *inward* and *outward* (see Fig. 2). Three spacings were evaluated for each configuration, namely $S/D = 1.8, 2$, and 2.5 .

2.2. Performance parameters

The hydrodynamic performance was quantified using dimensionless parameters, namely, torque coefficient, C_T (Eq. (1)), power coefficient C_p (Eq. (2)), and TSR, λ (Eq. (3)).

$$C_T = \frac{T}{0.5\rho A V_\infty^2} \quad (1)$$

$$C_p = \frac{P}{0.5\rho A V_\infty^3} \quad (2)$$

$$\lambda = \frac{\omega R}{V_\infty} \quad (3)$$

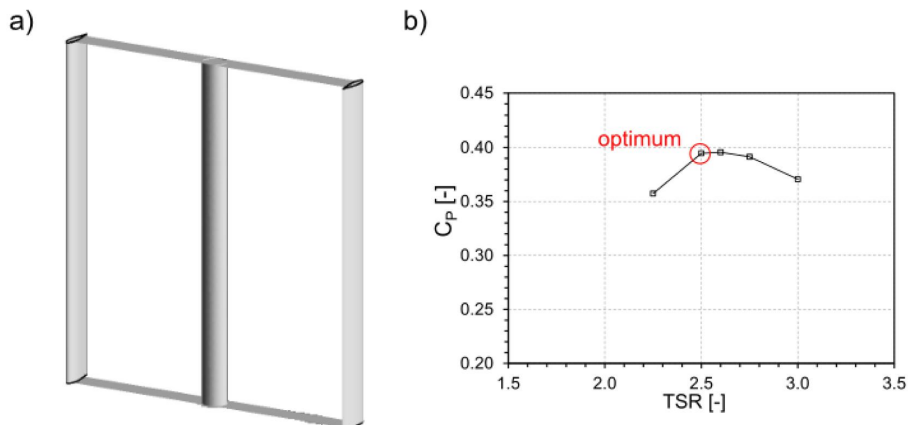


Fig. 1. A) schematic representation of the reference turbine and b) the performance curve.

Table 1

Geometrical characteristics of the test case turbine.

Dimension	Value
Number of blades [-]	2
Diameter, D [m]	2
Chord, c [m]	0.25
Rotor solidity (Nc/D), σ [-]	0.25
Airfoil	DU06W200
Blade-spoke connection [x/c]	0.25

Table 2

Test operating conditions.

Parameter	Value
Tip-speed ratio [-]	2.5
Rotational speed, ω [rad/s]	5
Work fluid [-]	water
Inflow speed, V_∞ [m/s]	2
Inflow temperature [$^{\circ}\text{C}$]	25
Inflow turbulence intensity [%]	0.24

In Eqs. (1)–(3), T is the torque on the shaft, ρ is the inflow density, A is the turbine frontal area, R is the radius of the turbine, V_∞ is the inflow velocity, P is the produced hydrodynamic power, and ω is the rotational speed. C_T was computed for each blade individually every timestep and then averaged over a full cycle to calculate the C_p . Although the flow field around the twin rotors is symmetrical, to minimize numerical errors, all performance parameters were averaged across both turbines. For example, (Eq. (4)) shows how the instantaneous global torque coefficient for the twin rotors system was calculated.

$$C_T(\theta) = \frac{C_{T,rotor1}(\theta) - C_{T,rotor2}(\theta)}{2} \quad (4)$$

3. Numerical model

3.1. CFD model

All simulations were conducted using the commercial CFD software ANSYS® FLUENT® version R20.2, using the Unsteady Reynolds-Averaged Navier-Stokes (U-RANS), two-dimensional, pressure-based solver. To minimize numerical diffusion errors, the second-order upwind convection scheme was utilized for the approximation of the convection/diffusion terms. An angular timestep $\Delta\theta$ of 0.4° was used for all simulations with the second-order implicit time advancing scheme.

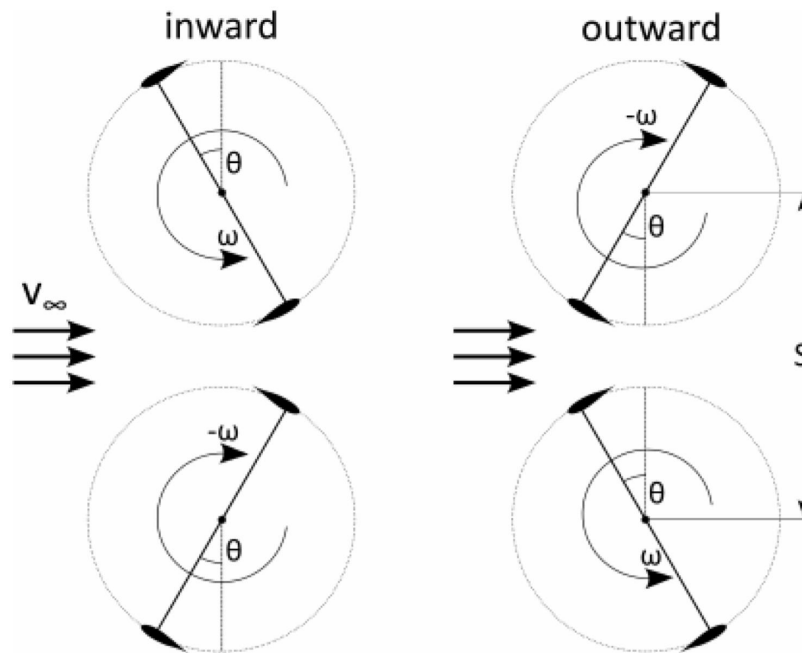


Fig. 2. Schematic representation of the inward (left) and outward (right) configurations.

The solution periodic convergence was assessed based on the average torque for a complete revolution. A threshold of 0.1% residual in the average torque was adopted, corresponding in approximately 25 revolutions depending on the operating point. Thus, only results of the last cycle were considered. The $k-\omega$ SST turbulence model [30] was used for turbulence closure.

The choice of the numerical schemes and turbulence model was

based on the experience of the authors on similar test cases [31,32]. The temporal and spatial discretization levels were selected to be in line with the comprehensive sensitivity studies conducted by the authors in [31,33], and particularly to be compliant with the dimensionless vorticity and Courant number thresholds proposed in [34]. The size of the computational domain size was adopted from previous studies on a single rotor [31], with some adjustments to accommodate the blockage

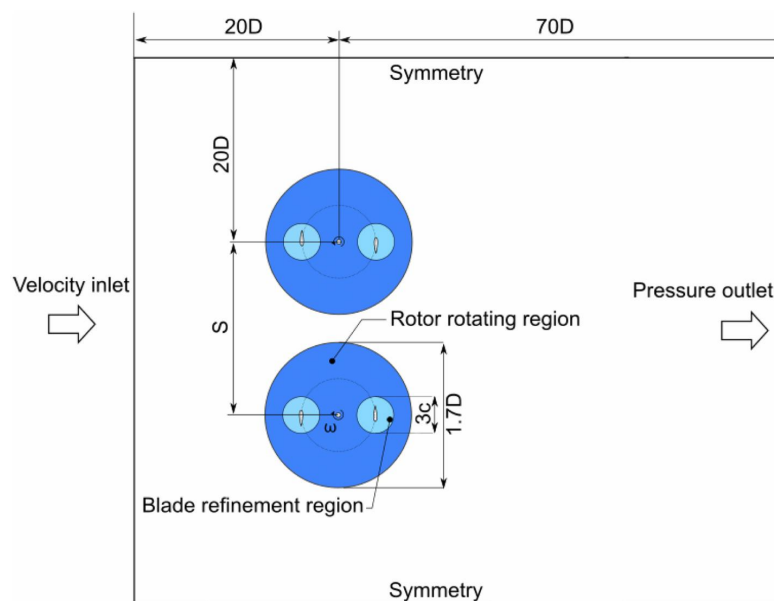


Fig. 3. Computational domain schematic.

effect due to the presence of two rotors. The domain was enlarged in the crosswise direction by a length S , which is the distance between the rotors. The domain consists of two rotating zones and a stationary zone. The rotating sub-domains and the stationary sub-domain were separated using non-conformal mesh where the *sliding mesh* technique was applied. The rotating domain was designed to be $1.7D$. The details of the domain dimensions as well as the boundary conditions are shown in Fig. 3.

Regarding the mesh, a non-structured mesh was adopted, similar to that applied in previous works of [31,33]. In this grid, the finest mesh was created in the zones surrounding the airfoils, using prismatic elements near the walls as shown in Fig. 4e. As required for the $k-\omega$ SST turbulence model, the first cell layer from the wall was sized to reach a dimensionless wall distance $y^+ < 1$. The rotating domains were divided into sections to ensure a gradual transition from the fine elements near the airfoil up to the coarse elements in the stationary domain. For the latter, two wake zones were created, where the mesh density increases gradually from the twin rotors until reaching the far field. The total element size of the created mesh is $\approx 1,500$ k. Table 3 reports the features of the mesh and Fig. 3 shows the details of the mesh created for the turbine pair.

3.2. Angle of attack sampling

Darrieus turbines are characterized by their complex flow field and unsteadiness as illustrated in Fig. 5. During their operation, the blades rotate with a cycloidal motion, causing intrinsic azimuthal variations of the aero/hydrodynamics, which result in complex flow phenomena such as dynamic stall, vortex shedding, and blade-wake interactions [35]. In addition, due to the blockage effect caused by the rotating blades, the inflow that approaches the Darrieus rotor deviates, and forms what is known as streamtube expansion. Hence, a simplified aerodynamic analysis could not achieve accurate estimations of the azimuthal variation of the kinematic quantities, e.g., angle of attack and blade local velocity.

The *LineAverage* method (LineAvg) was used to sample the local

Table 3

Details of the adopted mesh.

Parameter	Value
# elements in stationary domain	526,712
# elements in each rotating domain	490,136
# nodes on the blade	1440
# inflation layers on the airfoil	60

velocity field. This technique was first introduced for Horizontal-Axis Wind Turbines (HAWTs) to include the influence of the shedding and the trailing vorticity in the computed AoA [22]. Recently, it has been adapted for Darrieus turbines and validated against blade-resolved CFD and experimental measurements in [24,25]. In these works, it has been shown that the results computed by *LineAvg* could produce some discontinuities at low TSRs due to the interaction between the separation vortices and the sampling line downstream of the airfoil. In the present case, this is not an issue, given the stable operating conditions of the selected twin rotors.

In the *LineAvg*, the undistributed inflow velocity is sampled as an integral average along a closed line around the airfoil, centered in its aerodynamic center, i.e., approximately at 25% of the chord. In this way, the induced velocity components are levelled out during the averaging process, filtering out the effect of the profile bound circulation. Once the local velocities in the x - and y -direction have been sampled, they are combined to calculate the local inflow velocity, $V_{\infty, L}$, which is then combined with the rotor speed, ωR , to compute the AoA and relative velocity V_R :

$$V_{\infty, L} = \sqrt{V_x^2 + V_y^2} \quad (5)$$

$$\alpha = \tan^{-1} \left(\frac{-V_x \sin \theta + V_y \cos \theta}{-V_x \cos \theta - V_y \sin \theta - \omega R} \right) \quad (6)$$

$$V_R = \sqrt{(-V_x \sin \theta + V_y \cos \theta)^2 + (-V_x \cos \theta - V_y \sin \theta - \omega R)^2} \quad (7)$$

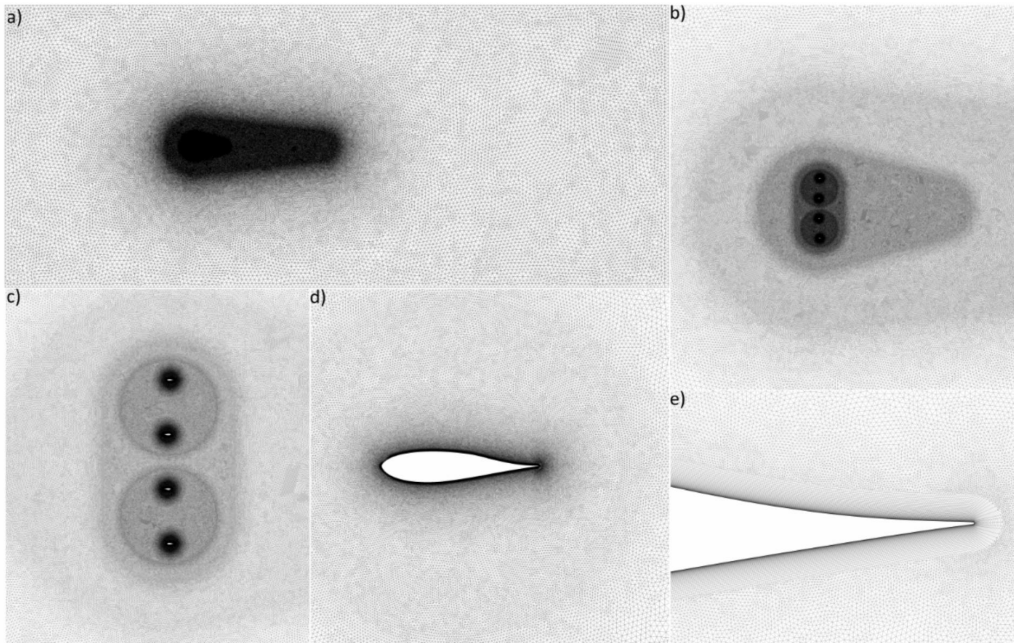


Fig. 4. Details of the adopted mesh: a) far field, b) near wake, c) rotating zones, d) blade, and e) trailing edge.

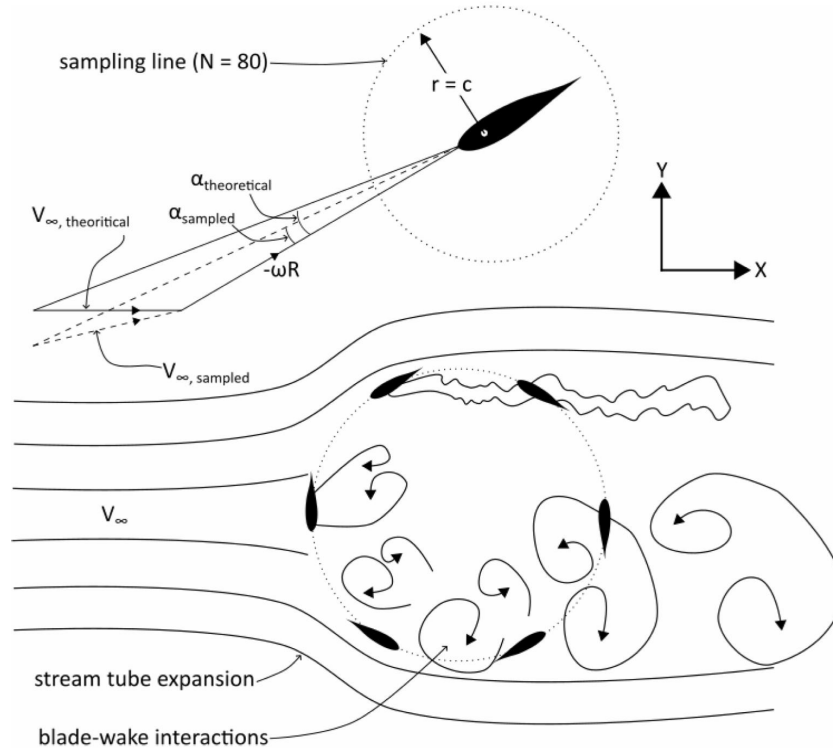


Fig. 5. Schematic representation of the Darrieus rotor flow field and the LinAvg AoA sampling method.

Being able to sample the local flow field, an induction factor (a) equivalent to that of the ubiquitous Blade Element Momentum Theory can be computed to represent the relative decrease in the inflow velocity between the freestream and the rotor domain:

$$a = \frac{V_{\infty} - V_{\infty,L}}{V_{\infty}} \quad (8)$$

In this study, a circular sampling line has been used, with a radius $r = 1c$ and 80 evenly distributed sampling points (see Fig. 5).

4. Results

In this section, the main findings of the current study are presented. First, the performance of the counter-rotating twin rotors is shown at different spacing distances, S , for both *inward* and *outward* configurations ($S = 1.8D$, $2D$, and $2.5D$), followed by a demonstration of the instantaneous torque production for a full revolution and local blade kinematic analysis. A series of comparisons in terms of angle of attack and induction factor between the single rotor and both configurations of the twin rotors are then presented to understand the flow regime that characterizes the twin rotors. Finally, coupling the kinematic data with its corresponding computed forces allowed the computation of the lift and drag data, which gave an insight into the physical mechanism that leads to an efficiency increase for twin Darrieus rotors.

4.1. Twin rotors performance

Fig. 6 shows the average C_p calculated for each setup at TSR 2.5, compared with that of the single rotor. The efficiency improvement percentage for the twin rotors with respect to the single rotor was calculated as in Eq. (8):

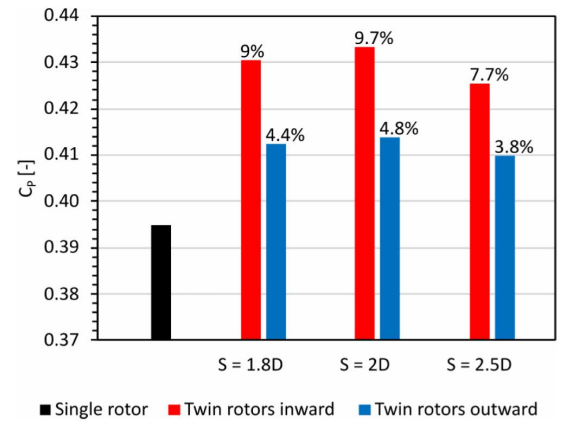


Fig. 6. Average power coefficient at TSR = 2.5 resulted from the twin rotors and their percentage increase compared to single rotor at different separating distances.

$$\text{Performance increase}[\%] = \frac{C_{p\text{TwinRotors}} - C_{p\text{SingleRotor}}}{C_{p\text{SingleRotor}}} \quad (9)$$

The results show that the separating distance of $S = 2D$ allowed achieving the highest power coefficient for both the *inward* and the *outward* setups, with efficiency improvement of 9.7% and 4.8%, respectively. Thus, this separating distance was considered for further investigations to analyze the mechanisms underlying such performance

enhancement.

To see the influence of TSR on the twin rotors performance, both *inward* and *outward* configurations were tested at different TSRs as reported in Fig. 7 along with the percentage improvement for each case. The results show that the optimum TSR for the twin rotors tends to shift to higher values compared to that of the single rotor, and the efficiency improvement increases with the increase in TSR. The *inward* setup achieves the highest power coefficient of 0.46 at TSR = 3, while the maximum power coefficient *outward* was 0.43 at TSR = 2.75, with percentages improvement of 21.8% and 12.8%, respectively. In general, comparing the optimum operating conditions of the twin rotors with that of the single rotor, the power coefficient improved by 16.1% and 8.7% for the *inward* and *outward*, respectively.

Fig. 8 shows the instantaneous torque coefficient and dimensionless normalized force computed for a single blade for the *inward* and *outward* twin rotors configurations compared with that of the single rotor at TSRs 2.5 and 3, respectively. The quadrant contribution was calculated by integrating the torque within the quarter and then dividing it over the total integral torque. The relative quadrant torque increase or decrease between the twin rotors and the single rotor was computed by multiplying the quadrant contribution by the quadrant percentage improvement. Table 4 reports the percentage power contribution for the single rotor and the rotor pairs with their relative increase/decrease of the twin rotors with respect to the single rotor.

For the *inward* configuration, which achieved higher power enhancements for both TSRs 2.5 and 3, the torque coefficient within the azimuth range of $0^\circ \leq \theta \leq 90^\circ$ is almost the same as that of the single rotor blade, showing a noticeable decrease in the quadrant contribution due to the overall increase in the torque production. The second and third azimuthal quarters, from azimuthal positions from $\theta = 90^\circ$ to $\theta = 270^\circ$ represent the area of mutual interaction, where the blades of the twin rotors are in the closest positions to the adjacent rotor. During the second quarter, almost at all azimuthal positions, the torque coefficient of the twin rotors is higher than that of the single rotor. Indeed, most of the performance enhancement is been achieved during this quarter, having a quadrant contribution of 63% at TSR = 2.5 and 90% at TSR = 3, compared to 60% and 85%, respectively, for the single rotor. The third quarter has a torque generation close to that of the single rotor, with a slight increase which could be better noted at TSR = 2.5 around $\theta \approx 225^\circ$. The final quarter shows almost the same performance as that of the single rotor for both TSRs 2.5 and 3.

Regarding the *outward* configuration, the areas of mutual interaction are the first and fourth quarters, i.e., $0^\circ \leq \theta \leq 90^\circ$ and $270^\circ \leq \theta \leq 360^\circ$.

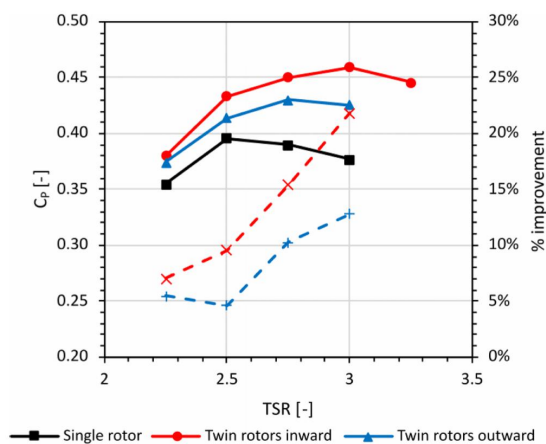


Fig. 7. Average power coefficient against TSR resulted from the twin rotors and their percentage increase compared to single rotor.

In the first quarter, most of the performance enhancement can be observed for both TSRs 2.5 and 3, with a peak torque slightly higher than that of the single rotor. On the other hand, during the second quarter, a small reduction in torque could be noticed at TSR = 2.5, while a slight increase is observed at TSR = 3. From $\theta = 180^\circ$ until $\theta = 360^\circ$ at both TSRs, the performance of the twin rotors is almost equal to that of the single rotor with a slight decrease in the torque coefficient at the last quadrant.

Overall, despite the higher performance enhancement achieved at TSR = 3 compared to that at TSR = 2.5, one can notice the resemblance in the quadrant increase/decrease in torque with respect to the single rotor, where the differences are mainly attributed to the influence of the TSR on quadrant torque contribution. It can be observed that for the *inward* setup, most of the enhancements occur during the second quarter, while they occur in the first quadrant for the *outward* setup.

4.2. Induction factor

Although the concept of “induction factor” comes from the one-dimensional momentum theory, it provides an intuitive way to analyze the flow modification and can be then used in a comparative sense to represent the relative decrease of the inflow speed through the turbine area and thus, an indication of the momentum flux that enters the rotor domain. Fig. 9 shows the variation of a of a single blade of the twin rotors compared to that of the single rotor for a full revolution at TSRs 2.5 and 3, respectively. The percentage increase/decrease was also calculated instantaneously and averaged for each azimuthal quarter. As can be seen for both TSRs, for the *inward* configuration, a was close to that of the single rotor during the first quarter, i.e., $0^\circ \leq \theta \leq 90^\circ$, with a decrease of 1.5% and 3.1%, for TSRs 2.5 and 3, respectively. Almost half of the second quarter, from $\theta \approx 90^\circ$ to $\theta \approx 135^\circ$, the induction factor was almost equal for both twin and single rotors. After that, a significant decrease a could be noticed for the *inward* twin rotors until matching again with the single rotor at $\theta \approx 315^\circ$. At TSR = 2.5, during the second and third quarters, a decreased by 3.2% and 7.6%, respectively. Closer reduction in a could also be seen at TSR = 3 with percentages of 3.8% in the second quarter and 6% in the third quarter. At both TSRs, no significant changes occur during the last quarter.

In the case of the *outward* setup, the decrease in a can only be observed in the second and last quarters for both TSRs with 2.4% and 2.5%, respectively, at TSR = 2.5, and 3.4% for both quarters at TSR = 3. Overall, with respect to the single rotor, the induction factor for the *outward* setup decreased by 2% at TSR = 2.5 and 6.9% at TSR = 3. On the other hand, for the *inward* setup it is decreased by 12.5% at TSR 2.5 and 13.6% at TSR = 3.

An interesting point is that the areas with an overall decrease in the induction factor do not always correspond to an increase in power coefficient (see Table 4). In the first quarter of the *inward* setup, the efficiency decreased, and yet the induction factor decreases. Conversely in the *outward* setup, the quarters where the enhancement of the power coefficient occurs are the same quarters showing an increase in the induction factor, and vice versa. The latter observation confirms that the efficiency improvement of the closely spaced rotors is not directly related to the increased volumetric flux through the rotor as suggested by Alexander et al. [16].

4.3. Blade hydrodynamics

Fig. 9 reports the changes in cyclic AoA (α) and local Reynolds number (Re_c), as well as the lift coefficient (c_l), C_L , for the *inward* and *outward* twin rotors compared to the standalone rotor. The data is shown for TSRs 2.5 and 3, along with the static airfoil data acquired using XFOIL [31]. It is important to note that the static airfoil data presented was obtained for the virtually cambered DU06W200 profile to account for the blade cycloidal motion [37].

At both TSRs, for the *inward* configuration, during the first quarter,

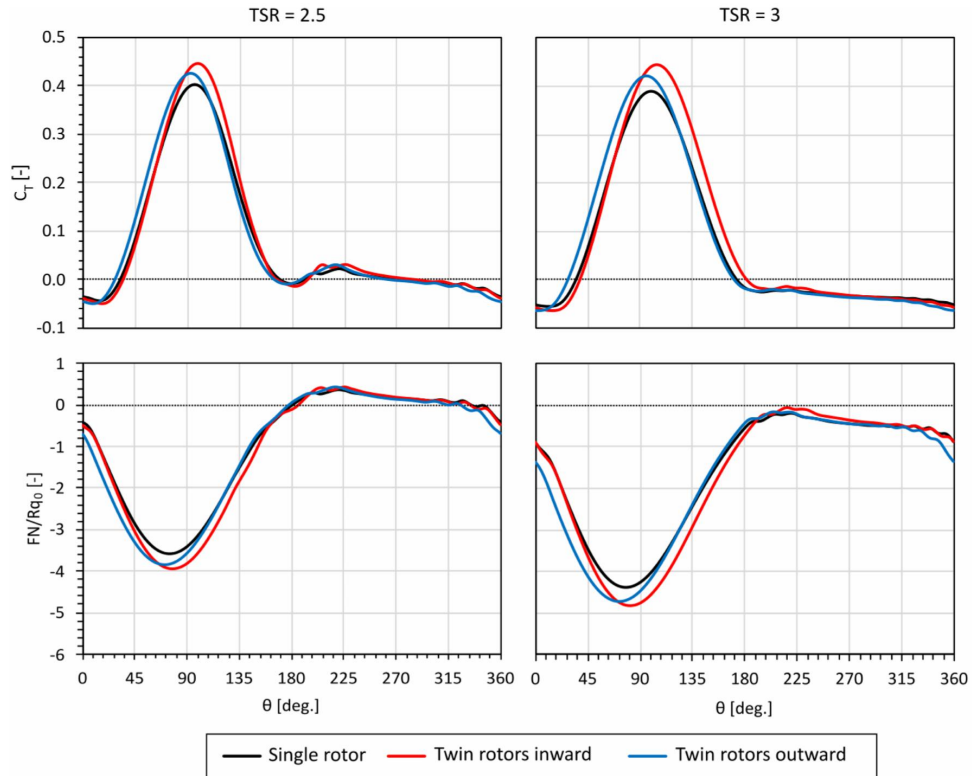


Fig. 8. Instantaneous torque coefficient and normalized normal force computed from the twin rotors at $S = 2D$ and TSRs 2.5 and 3 for inward and outward configurations compared to that of the single rotor.

Table 4
Power contribution for each cycle quarter and the corresponding twin rotors relative improvement/degradation.

θ	Single Rotor	Inward (% increase/decrease)	Outward (% increase/decrease)
TSR = 2.5			
0° to 90°	40.0%	34.7% (−1.7%)	47.3% (11.3%)
90° to 180°	60.1%	63.2% (9.6%)	53.8% (−3.5%)
180° to 270°	3.0%	4.7% (3.3%)	3.6% (0.9%)
270° to 360°	−3.1%	−2.6% (0.2%)	−4.7% (−2.7%)
TSR = 3			
0° to 90°	41.3%	31.7% (−2.0%)	50.3% (18.8%)
90° to 180°	85.2%	89.7% (25.5%)	76.5% (1.0%)
180° to 270°	−10.6%	−7.2% (1.2%)	−10.0% (−0.6%)
270° to 360°	−15.9%	−14.3% (−1.4%)	−16.8% (−3.3%)

the AoA variation matches that of the isolated rotor, the same observation can be made also for Re_c . After that, within the mutual interaction zone ($90^\circ \leq \theta \leq 270^\circ$), the blade experiences significant increases in AoA until $\theta = 180^\circ$, where $AoA = 0^\circ$, showing a complete suppression of the stream tube expansion effect. Thereafter, the AoA increases again above that of the single rotor after $\theta = 200^\circ$, until $\theta \approx 270^\circ$. Further observation within the mutual interaction zone is that Re_c , after matching with that of the single rotor until $\theta \approx 160^\circ$, decreases significantly up to $\theta \approx 270^\circ$. In the fourth quarter, i.e., $270^\circ \leq \theta \leq 360^\circ$, there are no relevant differences between the twin and isolated rotor.

For the *outward* configuration, in the first quarter, i.e., $0^\circ \leq \theta \leq 90^\circ$, which is a zone of mutual interaction in this setup, the AoA increases slightly compared to the single rotor, while Re_c decreases. The decrease

in the Re_c continued until $\theta \approx 160^\circ$, where it matches the single rotor values. Given that the flow in this setup is against the blade direction within the mutual interaction area, the increase in AoA is not as much as in the *inward* configuration. From $\theta \approx 90^\circ$ up to $\theta \approx 330^\circ$, the AoA experienced by the blade of the twin rotors almost matches that of the isolated rotor. Thereafter, it decreases marginally.

The computed lift coefficient displays clear hysteretic behavior, especially at TSR = 2.5, which is typical for Darrieus turbines due to dynamic effects, in particular dynamic stall. During the *pitch-up* phase, both twin rotor setups show slopes comparable to that of the single rotor, albeit with a slight offset towards higher lift values. This offset is more evident when compared to the static lift. As the blade AoA increases from 0° to $\approx -17^\circ$, the lift coefficient still increases despite the AoA exceeding the static stall limit. Both twin configurations are characterized by a maximum C_L value $\approx 9\%$ higher than that of the isolated rotor, in particular the *inward* one, which is then penalized afterwards. During the *pitch-down* phase in fact, there is a higher degree of fluctuation in lift variation against the AoA for the different configurations. These variations can be attributed to the AoA fluctuations between $\theta \approx 120^\circ$ and $\theta \approx 225^\circ$, which are related to the mutual interactions between the adjacent rotors. At TSR = 3, this effect is not present anymore, and all rotor configurations show higher lift in comparison to the static coefficients. The dynamic variation of lift at this TSR during the *pitch-down* phase in fact cannot be attributed to dynamic stall, but rather to other unsteady effects associated with elevated reduced frequencies, causing a delay in the effective AoA with respect to the one expected at the given azimuthal position. This phenomenon has been extensively analyzed on results obtained using the *LinAvg* for a Darrieus turbine in Ref. [24].

To see the kinematic characteristics of the twin Darrieus rotors from

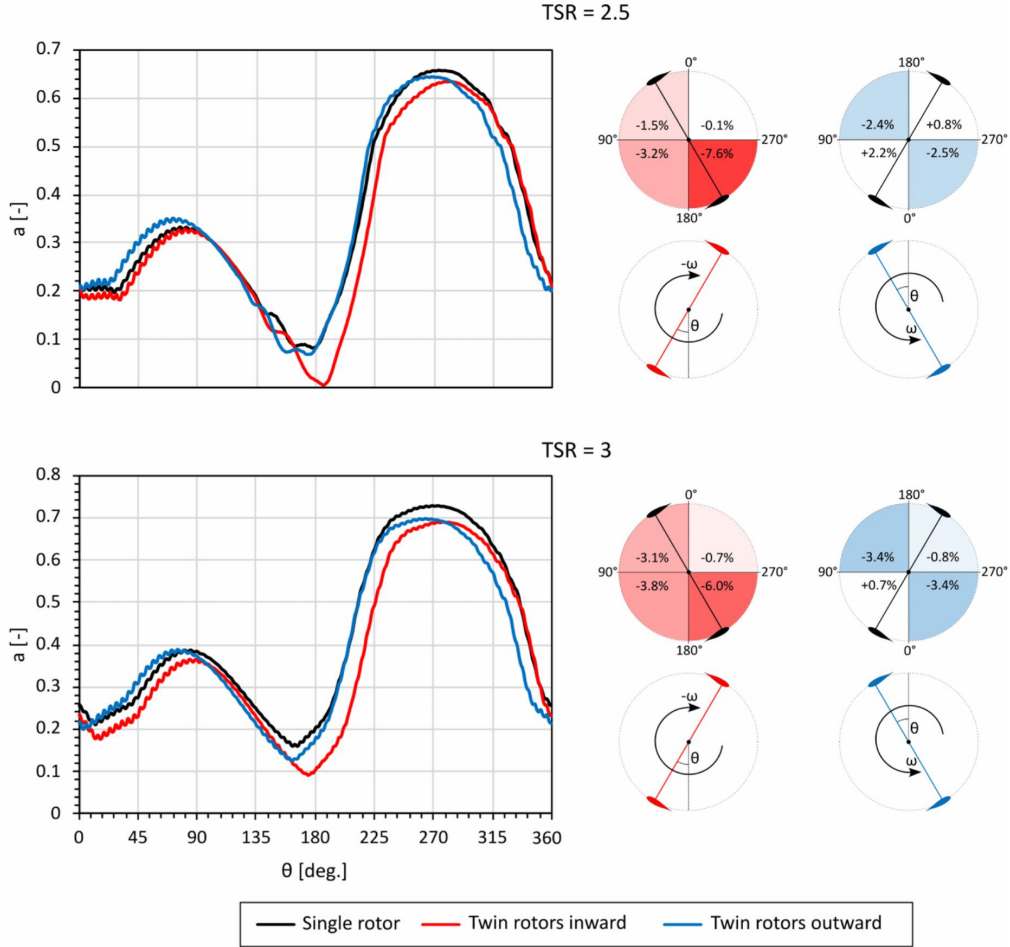


Fig. 9. Cyclic change in the induction factor for the a) inward and b) outward setups at $S = 2D$ compared to that of the single rotor.

a different perspective, Fig. 11 shows the contours of the dimensionless vorticity as well as the streamlines for both the twin rotors configurations and the single isolated rotor at $TSR = 3$. Upon examination of the streamlines, one can clearly notice the streamtube expansion in the isolated rotor. However, for both the *inward* and *outward* pairs, the mutual interaction between the rotors eliminates the deviation in inflow as marked in the black rectangle. For both cases, the blockage caused by the rotors allows the flow to be constrained in a narrow area, which leads to the suppression of the y-velocity components, making the flow almost straight. It is speculated that the latter effect is the main reason for the AoA increase observed previously (see Fig. 10).

Fig. 12 reports the computed lift coefficient against the azimuth position for both *inward* and *outward* configurations compared with that of the single rotor at $TSRs$ 2.5 and 3. The percentage increase of lift generation for each quarter was calculated as

$$\%C_L \text{ increase} = \frac{\int (|C_L|_{\text{TwinRotors}} - |C_L|_{\text{SingleRotor}}) d\theta}{\int |C_L|_{\text{SingleRotor}} d\theta} \quad (10)$$

The percentage increase of C_L for each quarter is close for both evaluated $TSRs$. For the *inward* configuration, the extra lift generation starts early at $\theta \approx 60^\circ$, it peaks at $\theta \approx 120^\circ$, and then decreases to match with that of the single rotor before the second rise at $\theta \approx 200^\circ$ until

matching again with the single rotor during the fourth azimuthal quarter. On the other hand, the *outward* starts with a lift coefficient higher than that of the single rotor until peaking at $\theta \approx 120^\circ$ and then matching with the single rotor up to the fourth quarter, where the blades enter again the zone of mutual interaction, and the lift coefficient starts to be higher than that of the single rotor from $\theta \approx 315^\circ$. For the *inward* setup, the highest percentage increase in lift occurs in the second and third quarters, while for the *outward* setup, it can only be found in the first quarter. Those quarters are also the same quarters with highest relative power coefficient enhancement (see Table 4).

4.4. Effect of azimuth shift

Given that the analyses presented in this work only consider twin rotors with blades perfectly aligned at the same corresponding azimuth positions, which is an ideal case that is hard to achieve in reality, an assessment has been conducted to evaluate the influence of azimuth shift between the two twin rotors, referred to as $\Delta\theta$. Fig. 13 shows the instantaneous azimuthal variation of torque coefficient for the inward setup at $TSR = 3$ for a perfectly aligned setup, i.e., $\Delta\theta = 0^\circ$ compared with that of $\Delta\theta = 90^\circ$. The choice of the latter azimuth shift is justified as it would be the widest azimuth shift that could be made for two-bladed Darrieus rotors. As can be seen, the torque coefficients of both cases

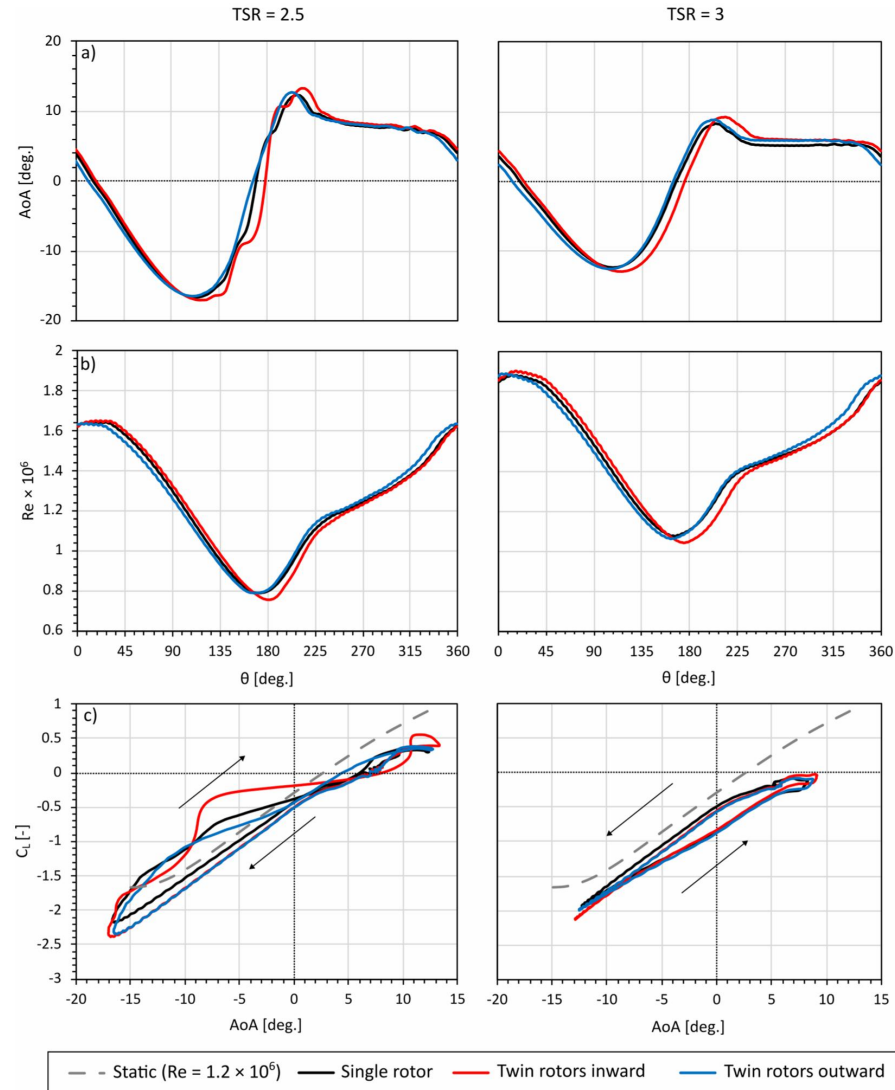


Fig. 10. Cyclic a) AoA and b) Re_c variations against azimuth position, and c) lift hysteresis loop for inward and outward twin rotors at TSRs 2.5 and 3 compared with the single rotor and the static lift.

perfectly match, which agrees with the findings of Jin et al. [36]. Accordingly, from the physical point of view, one can say that the twin rotors are seen by the flow field as spinning closed bodies regardless of the position of the blades, making the effect that the closely spaced rotors are imposing on the approaching flow field, which has been proven to be the main reason of the performance enhancement, uninfluenced by azimuth shift between the turbines.

5. Discussion

The present study quantitatively demonstrates how the efficiency of two closely-spaced counter-rotating Darrieus rotors increases compared to a single rotor. All results were obtained using a two-dimensional CFD U-RANS approach, which has been extensively validated for such an application against experimental results [39,40], and higher-fidelity approaches [41]. CFD settings have been optimized according to more

credited best practices for Darrieus turbines [31,42]. However, it is undisputed that this approach might suffer from some limitations, such as overestimating the power coefficient, as tip losses are not considered [43]. Additionally, the real blockage effect given by the two rotors will probably be lower than in 2D, as the flow can partially bypass the turbine in the blade span-wise direction. Therefore, the percentage increase in the power coefficient might actually be lower, as pointed out by Nazari et al. [44]. Complex flow structures and turbulence might be also not completely captured by a RANS approach. However, based on the discussed previous validations of this method and bearing in mind that the effects under investigation are related to the main flow behavior across the turbines, it is reasonable to assume that the results obtained in this study using U-RANS provide a consistent insight into the augmentation mechanism of twin rotors. Moreover, it should be also noted that a comparative approach has been adopted between the configurations, thus reducing the potential impact of some specific settings.

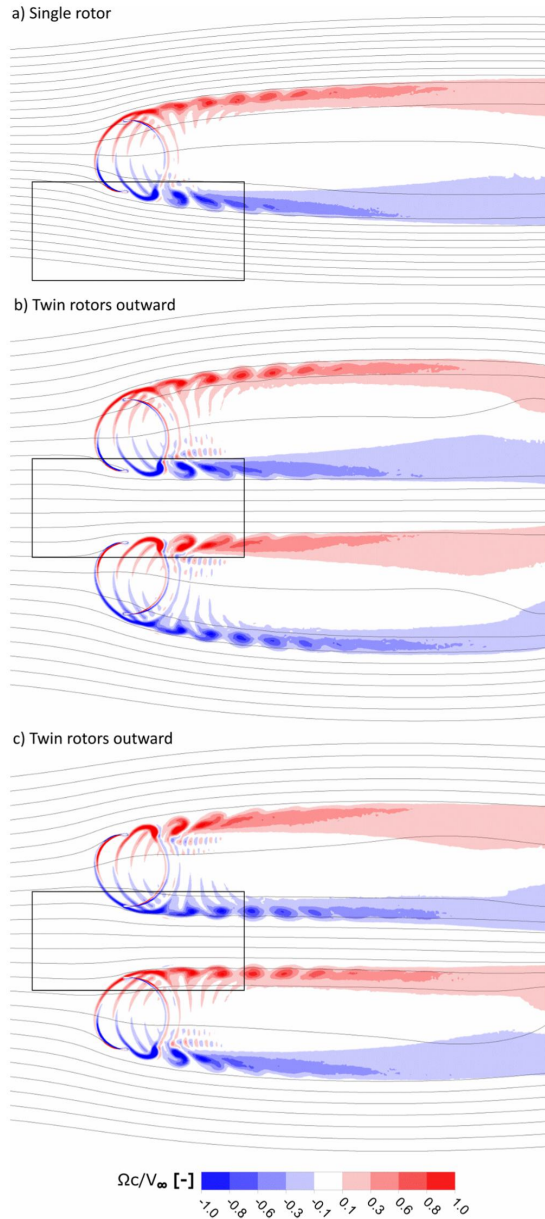


Fig. 11. DIMENSIONLESS contours and streamlines at TSR = 3 for a) single rotor, b) inward and c) outward twin rotors.

As a first step, it has been observed that the optimum operating point of twin rotors tends to increase to higher TSRs, in line with what was observed by Zanforlin et al. [15], Guilbot et al. [17], and Peng et al. [18]. The power augmentation mechanism was found to be the same at both TSR = 2.5 and TSR = 3, but the shift in the optimum operating conditions can be attributed to the more stable performance, resulting in a smoother variation of AoA, especially at the end of the first pitching-down phase and the beginning of the second pitching-up phase (i.e., $135^\circ < \theta < 225^\circ$ as shown in Fig. 10a). The quantitative analyses of the present study showed that this is mainly due to the higher blockage

effect experienced by the twin rotors at higher rotational speed, resulting in producing a more consistent increase in the lift coefficient, as illustrated in Fig. 12, where the inward configuration at TSR 2.5 showed a continuous increase up to $\theta \approx 160^\circ$.

With respect to previous studies, the present one employed flow field sampling to extract the real angle of attack (AoA) and compute the effective lift and drag coefficients of the blade. The results revealed two key effects that arise due to the close deployment of turbines. First, the decrease in the axial induction factor indicated an increase in the momentum entering the turbine domain. This finding aligns with Alexander et al.'s [16] observation, as they suggested this effect to be the main source of power augmentation. However, differently from that study, it was found that the azimuth positions where the induction factor decreases are not always the same positions with enhanced efficiency. Secondly, the suppression of the cross-stream velocity component resulted in a reorientation of the relative velocity component, which was found to significantly enhance the lift generated by the blades. This latter finding, which Zanforlin et al. [15] identified as the main source of power increase, was quantified in this study to gain a better understanding of the power augmentation mechanism of twin rotors.

The results of the present work suggest that the power augmentation mechanism of the twin rotors follows a specific sequence. Firstly, the presence of two rotors generates a blockage effect in the mutual interaction zone. As a result, the stream tube expansion is eliminated, suppressing the cross-stream velocity component. Secondly, the straightening of the inflow increases the angle of attack (AoA) within the mutual interaction area, which increases the lift. Indeed, the change in the direction of the relative velocity also makes it become more oriented towards generating higher tangential forces. This is due to the fact that the lift component, which has now been reoriented to align with the increased AoA, is now also more aligned with the tangential force component. As a result, the mechanism is capable of enhancing both the lift and tangential force components, ultimately leading to an overall increase in power output. These effects are more pronounced for the inward configuration because the mutual interaction region lies in the area where most of the power generation takes place.

It is important to note that, based on the results presented in this paper, the power augmentation mechanism is mainly driven by the AoA increase. In fact, the latter observation could be useful in the design optimization of closely spaced Darrieus rotors starting from the airfoil selection up to the implementation of further power augmentation techniques, i.e., pitch control [27] and adding flow deflectors [14]. The findings could also be useful in validating and tuning low-fidelity models for closely spaced Darrieus turbines such as those in Refs. [6,8]. Future studies will be devoted at further investigating these aspects.

6. Conclusions

The flow physics around a pair of hydrokinetic Darrieus turbines have been investigated using two-dimensional CFD and a method to extrapolate the angle of attack from the solved flow field. This second method represented the key enabler to understand the impact of flow modifications at the blade level.

First, different spacings between the rotors have been investigated for both the *inward* and *outward* configurations: 1.8D, 2D, and 2.5D. The optimum spacing was found to be 2D for both *inward* and *outward* setups. With this spacing, the maximum power coefficient is however reached at different tip-speed ratios. Overall, it can be noted that the twin rotors operate at an optimum TSR higher than that of the single rotor. More specifically, the single rotor optimum TSR is 2.5, which corresponds to a C_p of 0.4. On the other hand, the *inward* setup showed the highest C_p of 0.46 at TSR = 3, while the *outward* setup had a maximum C_p of 0.43 at TSR = 2.75, with improvements of 16.1% and 8.7%, respectively.

Detailed results of the blade local flow field at both TSRs 2.5 and 3 showed an increase in the AoA within the area of mutual flow

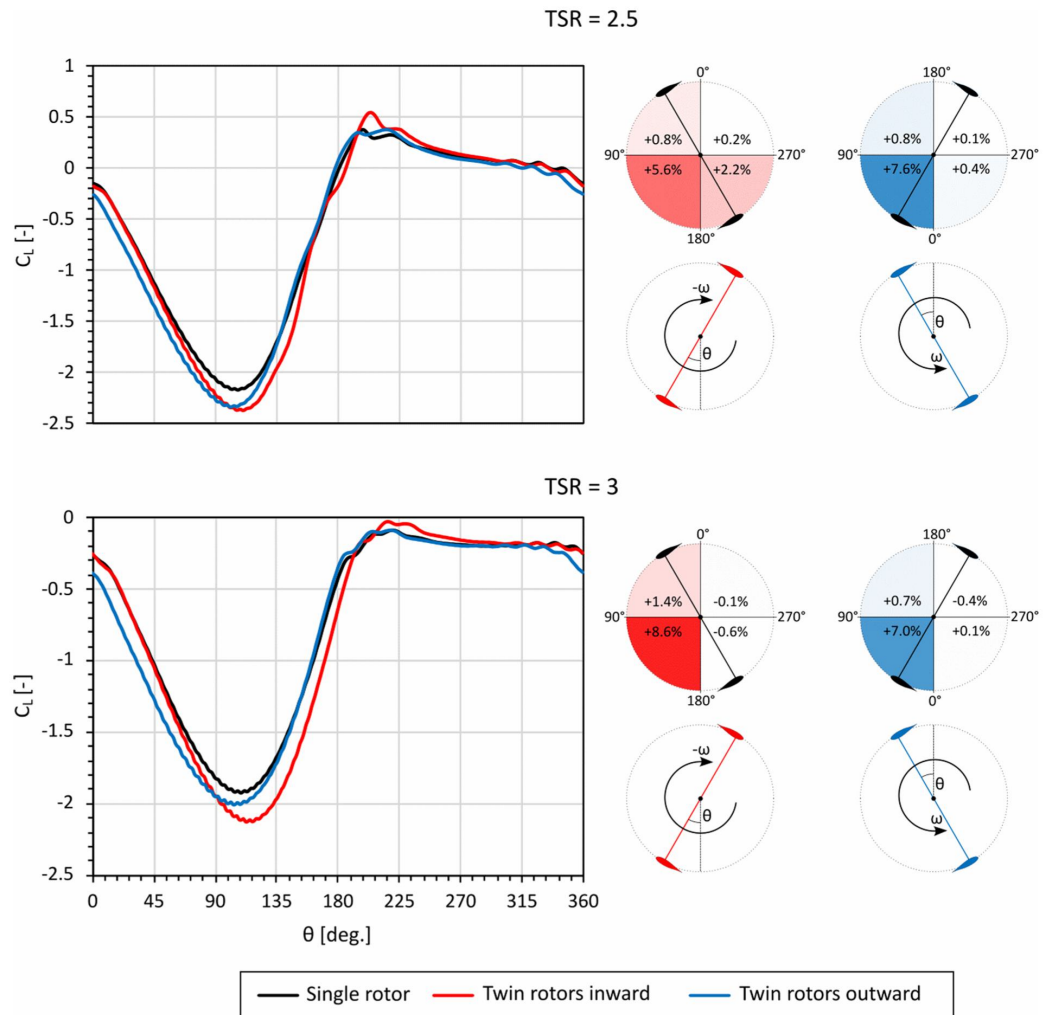


Fig. 12. Lift coefficient results on a single blade against azimuth position of a) inward b) outward configurations at $S = 2D$ compared with that of the single rotor for one revolution.

interactions between the two adjacent rotors. It can be concluded that the blockage resulting from the flow field of the two closely spaced rotors forces the inflow to be straightened and suppresses the stream tube expansion effect. Thus, it causes a change in the relative velocity direction, which increases the AoA. This increase in the AoA allows the blade to generate higher lift compared to the single rotor. In the *inward* configuration, the blade experiences this increase in the AoA during the pitch-down phase, producing more lift than that of the isolated rotor. On the other hand, for the outward configuration, the blade experiences this AoA increase during the pitch-up phase, which causes a slight but not insignificant increase in the lift compared to the isolated rotor.

The following points summarize the main findings of this work:

- Deploying Darrieus rotors closely in an inline layout creates a blockage in the area between the rotors, forcing the inflow to be straightened, and eliminating the streamtube expansion effect between the rotors;
- The elimination of the streamtube expansion allows more momentum flux to enter the rotor domain. This effect has been

quantified by computing the induction factor using the sampled flow field, showing that it has no direct relation with the increase in power coefficient;

- The change in the inflow direction creates a favorable increase in the AoA, which causes an increase in the lift force. The instantaneous lift coefficient has been computed in this work and is thought to be the main source of performance enhancement;
- The latter effects are more pronounced at higher rotational speeds. Therefore, the optimum power coefficient tends to shift towards higher TSRs in both the *inward* and *outward* setups;
- At $TSR = 3$, the lift generation increased by 8.6% in the second quarter of the *inward* setup, which corresponds to a 25.5% relative increase in the power coefficient. For the *outward* setup, it increased by 7% in the first quarter which corresponds to 18.8% in the power coefficient;
- The azimuth shift between the two adjacent rotors does not have any significant influence on effects leading to performance enhancement.

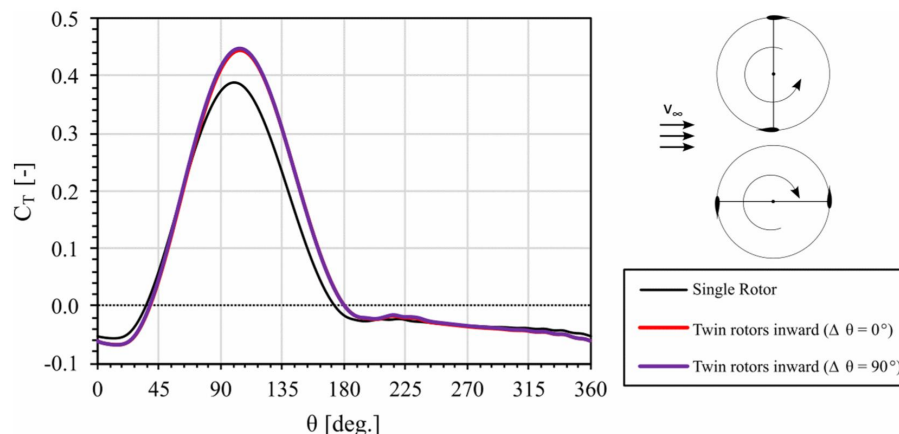


Fig. 13. Instantaneous torque coefficient variation against azimuth position for two twin rotors cases with different blade azimuth shifts, $\Delta\theta = 0^\circ$ and 90° , compared with that of the single rotor.

CRedit authorship contribution statement

Omar S. Mohamed: Conceptualization, Methodology, Software, Validation, Formal analysis, Investigation, Data curation, Writing – original draft, Visualization. **Pier Francesco Melani:** Methodology, Data curation, Writing – review & editing. **Francesco Balduzzi:** Conceptualization, Methodology, Writing – review & editing. **Giovanni Ferrara:** Supervision, Resources. **Alessandro Bianchini:** Conceptualization, Methodology, Investigation, Data curation, Writing – review & editing, Supervision, Project administration, Funding acquisition.

Declaration of Competing Interest

The authors declare that they have no known competing financial interests or personal relationships that could have appeared to influence the work reported in this paper.

Data availability

No data was used for the research described in the article.

References

- [1] Kumar R, Raahemifar K, Fung AS. A critical review of vertical axis wind turbines for urban applications. *Renew Sustain Energy Rev* Jun. 2018;89:281–91. <https://doi.org/10.1016/j.rser.2018.03.033>.
- [2] Paulsen US, Madsen HA, Hattel JH, Baran I, Nielsen PH. Design optimization of a 5 MW floating offshore vertical-axis wind turbine. *Energy Procedia* Jan. 2013;35:22–32. <https://doi.org/10.1016/j.egypro.2013.07.155>.
- [3] Tjiu W, Marnoto T, Mat S, Ruslan MH, Sopian K. Darrieus vertical axis wind turbine for power generation II: challenges in HAWT and the opportunity of multi-megawatt Darrieus VAWT development. *Renew Energy* Mar. 2015;75:560–71. <https://doi.org/10.1016/j.renene.2014.10.039>.
- [4] Fadil J, Soedibyo, Ashari M. Performance comparison of vertical axis and horizontal axis wind turbines to get optimum power output. In: 2017 15th International Conference on Quality in Research (QIR): International Symposium on Electrical and Computer Engineering, Jul. 2017. p. 429–33. doi: 10.1109/QIR.2017.8168524.
- [5] Dabiri JO. Potential order-of-magnitude enhancement of wind farm power density via counter-rotating vertical-axis wind turbine arrays. *J Renew Sustain Energy* Jul. 2011;3(4):043104. <https://doi.org/10.1063/1.3608170>.
- [6] Vergaerde A, De Troyer T, Standaert L, Kluczevska-Bordier J, Pitance D, Immas A, et al. Experimental validation of the power enhancement of a pair of vertical-axis wind turbines. *Renew Energy* 2020;146:181–7.
- [7] Pameix N, Fuchs R, Immas A, Silvert F. Efficiency improvement of vertical-axis wind turbines with counter-rotating lay-out. In: Proceedings of EWEA 2016.
- [8] Cacciali L, Battisti L, Dell'Anna S. Multi-array design for hydrokinetic turbines in hydropower canals. *Energies* Jan. 2023;16:5. <https://doi.org/10.3390/en16052279>.
- [9] Li S, Li Ye, Yang C, Wang Q, Zhao B, Li D, et al. Experimental investigation of solidity and other characteristics on dual vertical axis wind turbines in an urban environment. *Energy Convers Manage* 2021;229:113689. <https://doi.org/10.1016/j.enconman.2020.113689>.
- [10] Müller S, Muhawenimana V, Wilson CAME, Ouro P. Experimental investigation of the wake characteristics behind twin vertical axis turbines. *Energy Convers Manage* Nov. 2021;247:114768. <https://doi.org/10.1016/j.enconman.2021.114768>.
- [11] Mohamed OS, Ibrahim A, El Baz AMR. CFD Investigation of the multiple rotors Darrieus type turbine performance. In: Presented at the ASME Turbo Expo 2019: Turbomachinery Technical Conference and Exposition, Nov. 2019. doi: 10.1115/GT2019-91491.
- [12] Zheng H-D, Zheng XY, Zhao SX. Arrangement of clustered straight-bladed wind turbines. *Energy* Jun. 2020;200:117563. <https://doi.org/10.1016/j.energy.2020.117563>.
- [13] Sahebzaadeh S, Rezaeiha A, Montazeri H. Towards optimal layout design of vertical-axis wind-turbine farms: double rotor arrangements. *Energy Convers Manage* Dec. 2020;226:113527. <https://doi.org/10.1016/j.enconman.2020.113527>.
- [14] Zhao P, Jiang Y, Liu S, Stoesser T, Zou L, Wang K. Investigation of fundamental mechanism leading to the performance improvement of vertical axis wind turbines by deflector. *Energy Convers Manage* Nov. 2021;247:114680. <https://doi.org/10.1016/j.enconman.2021.114680>.
- [15] Zanforlin S, Nishino T. Fluid dynamic mechanisms of enhanced power generation by closely spaced vertical axis wind turbines. *Renew Energy* Dec. 2016;99:1213–26. <https://doi.org/10.1016/j.renene.2016.08.015>.
- [16] Alexander AS, Santhanakrishnan A. Mechanisms of power augmentation in two side-by-side vertical axis wind turbines. *Renew Energy* Apr. 2020;148:600–10. <https://doi.org/10.1016/j.renene.2019.10.149>.
- [17] Guilbot M, Barre S, Balarac G, Bonamy C, Guillaud N. A numerical study of Vertical Axis Wind Turbine performances in twin-rotor configurations. *J Phys: Conf Ser* Sep. 2020;1618(5):052012. <https://doi.org/10.1088/1742-6596/1618/5/052012>.
- [18] Peng HY, Han ZD, Liu HJ, Lin K, Lam HF. Assessment and optimization of the power performance of twin vertical axis wind turbines via numerical simulations. *Renew Energy* Mar. 2020;147:43–54. <https://doi.org/10.1016/j.renene.2019.08.124>.
- [19] Miller MA, Duvvuri S, Hultmark M. Solidity effects on the performance of vertical-axis wind turbines. *Flow* 2021;1:E9. <https://doi.org/10.1017/flo.2021.9>.
- [20] Rahimi H, Schepers JG, Shen WZ, Garcia NR, Schneider MS, Micallef D, et al. Evaluation of different methods for determining the angle of attack on wind turbine blades with CFD results under axial inflow conditions. *Renew Energy* 2018;125:866–76.
- [21] Bianchini A, Balduzzi F, Ferrara G, Persico G, Dossena V, Ferrari L. A critical analysis on low-order simulation models for darrieus vawts: how much do they pertain to the real flow? *J Eng Gas Turbines Power* Sep. 2018;141(1). <https://doi.org/10.1115/1.4040851>.
- [22] Jost E, Klein L, Leipprand H, Lutz T, Krämer E. Extracting the angle of attack on rotor blades from CFD simulations. *Wind Energy* 2018;21(10):807–22. <https://doi.org/10.1002/we.2196>.
- [23] Elsakka MM, Ingham DB, Ma L, Pourkashanian M. CFD analysis of the angle of attack for a vertical axis wind turbine blade. *Energy Convers Manage* Feb. 2019;182:154–65. <https://doi.org/10.1016/j.enconman.2018.12.054>.
- [24] Melani PF, Balduzzi F, Ferrara G, Bianchini A. How to extract the angle attack on airfoils in cycloidal motion from a flow field solved with computational fluid dynamics? Development and verification of a robust computational procedure. *Energy Convers Manage* Nov. 2020;223:113284. <https://doi.org/10.1016/j.enconman.2020.113284>.
- [25] Mohamed OS, Elbaz AMR, Bianchini A. A better insight on physics involved in the self-starting of a straight-blade Darrieus wind turbine by means of two-dimensional

- Computational Fluid Dynamics. *J Wind Eng Ind Aerodyn* Nov. 2021;218:104793. <https://doi.org/10.1016/j.jweia.2021.104793>.
- [26] Balduzzi F, Melani PF, Soraperra G, Brighenti A, Battisti L, Bianchini A. Some design guidelines to adapt a Darrieus vertical axis turbine for use in hydrokinetic applications. *E3S Web Conf* 2021;312:08017. <https://doi.org/10.1051/e3sconf/202131208017>.
- [27] Melani PF, Balduzzi F, Ferrara G, Bianchini A. Development of a desmodromic variable pitch system for hydrokinetic turbines. *Energy Conver Manage* Dec. 2021; 250:114890. <https://doi.org/10.1016/j.enconman.2021.114890>.
- [28] Melani PF, Balduzzi F, Ferrara G, Bianchini A. Tailoring the actuator line theory to the simulation of Vertical-Axis Wind Turbines. *Energy Conver Manage Sep. 2021; 243:114422*. <https://doi.org/10.1016/j.enconman.2021.114422>.
- [29] Mohamed OS, Melani PF, Balduzzi F, Ferrara G, Bianchini A. An insight on the key factors influencing the accuracy of the actuator line method for use in vertical-axis turbines: Limitations and open challenges. *Energy Conver Manage Oct. 2022;270: 116249*. <https://doi.org/10.1016/j.enconman.2022.116249>.
- [30] Two-equation eddy-viscosity turbulence models for engineering applications | AIAA Journal. <https://arc.aiaa.org/doi/abs/10.2514/3.12149?journalCode=ajaa> (accessed Nov. 13, 2022).
- [31] Balduzzi F, Bianchini A, Maleci R, Ferrara G, Ferrari L. Critical issues in the CFD simulation of Darrieus wind turbines. *Renew Energy Jan. 2016;85:419–35*. <https://doi.org/10.1016/j.renene.2015.06.048>.
- [32] Campobasso MS, Yan M, Bonfiglioli A, Gigante FA, Ferrari L, Balduzzi F, et al. Low-speed preconditioning for strongly coupled integration of Reynolds-averaged Navier-Stokes equations and two-equation turbulence models. *Aerosp Sci Technol* 2018;77:286–98.
- [33] Ibrahim AA, Elbaz AMR, Melani PF, Mohamed OS, Bianchini A. Power augmentation of Darrieus wind turbine blades using trapped vortex cavity. *J Wind Eng Ind Aerodyn Apr. 2022;223:104949*. <https://doi.org/10.1016/j.jweia.2022.104949>.
- [34] Balduzzi F, Bianchini A, Ferrara G, Ferrari L. Dimensionless numbers for the assessment of mesh and timestep requirements in CFD simulations of Darrieus wind turbines. *Energy Feb. 2016;97:246–61*. <https://doi.org/10.1016/j.energy.2015.12.111>.
- [35] Wang Y, Sun XJ, Zhu B, Zhang HJ, Huang DG. Effect of blade vortex interaction on performance of Darrieus-type cross flow marine current turbine. *Renew Energy Feb. 2016;86:316–23*. <https://doi.org/10.1016/j.renene.2015.07.089>.
- [36] Drela M. XFOIL: An Analysis and design system for low reynolds number airfoils. In: Conference on Low Reynolds Number Airfoil Aerodynamics, University of Notre Dame, Jun. 1989. doi: 10.1007/978-3-642-84010-4_1.
- [37] Rainbird JM, Bianchini A, Balduzzi F, Peiró J, Graham JMR, Ferrara G, et al. On the influence of virtual camber effect on airfoil polars for use in simulations of Darrieus wind turbines. *Energy Conver Manage* 2015;106:373–84.
- [38] Bianchini A, Balduzzi F, Bachant P, Ferrara G, Ferrari L. Effectiveness of two-dimensional CFD simulations for Darrieus VAWTs: a combined numerical and experimental assessment. *Energy Conver Manage Mar. 2017;136:318–28*. <https://doi.org/10.1016/j.enconman.2017.01.026>.
- [39] Rezaeiha A, Montazeri H, Blocken B. On the accuracy of turbulence models for CFD simulations of vertical axis wind turbines. *Energy Aug. 2019;180:838–57*. <https://doi.org/10.1016/j.energy.2019.05.053>.
- [40] Syawitri TP, Yao YF, Chandra B, Yao J. Comparison study of URANS and hybrid RANS-LES models on predicting vertical axis wind turbine performance at low, medium and high tip speed ratio ranges. *Renew Energy May 2021;168:247–69*. <https://doi.org/10.1016/j.renene.2020.12.045>.
- [41] Rezaeiha A, Montazeri H, Blocken B. Towards accurate CFD simulations of vertical axis wind turbines at different tip speed ratios and solidities: Guidelines for azimuthal increment, domain size and convergence. *Energy Conver Manage Jan. 2018;156:301–16*. <https://doi.org/10.1016/j.enconman.2017.11.026>.
- [42] Balduzzi F, Drofelnik J, Bianchini A, Ferrara G, Ferrari L, Campobasso MS. Darrieus wind turbine blade unsteady aerodynamics: a three-dimensional Navier-Stokes CFD assessment. *Energy Jun. 2017;128:550–63*. <https://doi.org/10.1016/j.energy.2017.04.017>.
- [43] Nazari S, Zamani M, Moshizi SA. Comparison between two-dimensional and three-dimensional computational fluid dynamics techniques for two straight-bladed vertical-axis wind turbines in inline arrangement. *Wind Engineering. Dec 2018;42: 647–64*. <https://doi.org/10.1177/0309524X18780384>.

Further reading

- [38] Jin G, Zong Z, Jiang Y, Zou L. Aerodynamic analysis of side-by-side placed twin vertical-axis wind turbines. *Ocean Eng Aug. 2020;209:107296*. <https://doi.org/10.1016/j.oceaneng.2020.107296>.



Omar Sherif Mohamed

Università degli Studi di Firenze,
Via di Santa Marta 3,
Firenze 50139, Italy
e-mail: omar.sherif@unifi.it

Pier Francesco Melani

Università degli Studi di Firenze,
Via di Santa Marta 3,
Firenze 50139, Italy
e-mail: pierfrancesco.melani@unifi.it

Giuseppe Soraperra

HE-PowerGreen s.r.l.,
Via Solteri 74,
Trento 38121, Italy
e-mail: gsoraperra@he-powergreen.it

Alessandra Brighenti

HE-PowerGreen s.r.l.,
Via Solteri 74,
Trento 38121, Italy
e-mail: amministrazione@he-powergreen.it

Francesco Balduzzi

Università degli Studi di Firenze,
Via di Santa Marta 3,
Firenze 50139, Italy
e-mail: francesco.balduzzi@unifi.it

Alessandro Bianchini

Università degli Studi di Firenze,
Via di Santa Marta 3,
Firenze 50139, Italy
e-mail: alessandro.bianchini@unifi.it

Selection Criteria for Rotor Number and Rotational Direction in Arrays of Closely Spaced Darrieus Turbines

The efficiency gains observed in a pair of closely spaced Darrieus turbines suggest the deployment of multiple turbines as an appealing solution for wind and, particularly, hydrokinetic applications, where the inflow direction is constant. The present study develops some design guidelines for closely spaced hydrokinetic Darrieus turbines by analyzing the trends of both power augmentation and wake development within arrays of multiple rotors, including not only an even number of rotors, which is the usual case in literature, but also an odd one. The analysis is carried out by means of two-dimensional computational fluid dynamics simulations and includes not only the assessment of instantaneous blade forces but also locally sampled flow fields past each blade that allowed the reconstruction of dynamic polar data, contributing to a more comprehensive understanding of the physical mechanisms at play in such compact setups. The study demonstrates a consistent power augmentation mechanism across different layouts, even in the case of an odd number of rotors. This enhancement originates from flow blockage in the mutual interaction areas, favorably altering the inflow angle and subsequently increasing the angle of attack and lift generation. While this mechanism aligns with previous observations on arrays of counter-rotating pairs, its application to multiple turbines introduces complexities due to potential asymmetries in inflow, leading to an uneven power enhancement across turbines within the array. The identified efficiency improvement pattern suggests that maximizing leeward mutual interactions within an array of multiple Darrieus turbines would enhance the overall efficiency of the setup. [DOI: 10.1115/1.4067007]

Keywords: VAWTs, multi-rotors, hydrokinetic turbines, CFD, turbine arrays

Introduction

While horizontal-axis turbines are generally considered to be more efficient and technologically mature, representing the state of the art in the wind energy industry [1], vertical-axis turbines offer unique advantages in hydrokinetic and tidal applications that could be beneficial in specific contexts. One of these advantages is the ability to efficiently deploy multiple rotors in dense layouts, exploiting the beneficial flow interactions between them. In fact, several studies have found that deploying two or more closely spaced Darrieus turbines can improve the overall efficiency of the array compared to the same number of standalone turbines. The first experimental proof of this concept was demonstrated by Dabiri [2], who conducted field tests on closely spaced pairs of Vertical Axis Wind Turbines (VAWTs) and found that clustering Darrieus turbines significantly boosts the power coefficient of the array by

up to 10% compared to individual turbines operating independently. More recently, Vergaerde et al. [3] investigated through wind tunnel tests a pair of closely spaced counter-rotating VAWTs, showing a relative power increase of 16% compared to an isolated rotor. Li et al. [4] conducted an experimental investigation on twin Darrieus rotors in an urban environment and observed similar performance improvements. Müller et al. [5] experimentally analyzed the wake behind twin Darrieus hydrokinetic turbines in both corotating and counter-rotating schemes. They highlighted that the wake region of the corotating pair expands more in lateral and vertical directions compared to the two potential configurations of the counter-rotating pair: leeward (turbine blades moving along with the flow in the mutual interaction area) and windward (turbine blades moving against the flow in the mutual interaction area).

Alongside experimental investigations, numerical studies, particularly employing computational fluid dynamics (CFD), have been conducted to optimize the array design or understand the physical mechanisms behind the efficiency enhancement observed in the experiments. Zanforlin et al. [6] used 2D Reynolds-averaged Navier-Stokes (RANS) CFD simulations to suggest two factors that lead to increased power production, i.e., (a) a favorable

Turbo Expo, June 24–28, 2024, GT2024.

Contributed by the Fluids Engineering Division of ASME for publication in the JOURNAL OF FLUIDS ENGINEERING. Manuscript received August 21, 2024; final manuscript received October 16, 2024; published online November 21, 2024. Assoc. Editor: Ryoichi S. Amano.

deviation in inflow velocity, and (b) a narrowing of the wind wake downstream that results in an increase in available momentum flux. Alexander et al. [7] expanded this analysis by noting that placing turbines in close proximity increases the volume of air flowing into them due to the blockage effect caused by the adjacent rotors. In a recent study [8], the authors of this work carried out a dedicated analysis of the underlying physics of a pair of counter-rotating Darrieus turbines. The results revealed that the in-line arrangement of a pair of counter-rotating turbines induces a blockage between them, which reduces the inflow deflection and leads to an increase in the angle of attack. The latter, if the operating tip-speed ratio is high enough, leads to an increase in the lift force, ultimately improving the performance of the turbines.

Regarding the array layout optimization, Mohamed et al. [9] found that closely spaced rotors arranged in-line without staggering exhibit the highest efficiency. Subsequently, Zheng et al. [10] corroborated this, affirming that the in-line configuration has higher efficiency than staggered setups. Furthermore, Zhao et al. [11] concluded that incorporating a flow deflector between counter-rotating turbines can increase the overall power coefficient by up to 23%. However, this modification increases sensitivity to the inflow direction, thereby impacting actual energy conversion.

The multirotor Darrieus turbines concept has also attracted interest of the industrial sector for various applications such as floating offshore wind turbines (FOWT), e.g., Nenuphar's Twinfloat [12], and hydrokinetic turbines (HKT). The main drawback of this setup is that the performance becomes more sensitive to the inflow direction, losing the omni-directionality typical of Darrieus turbines. This is not an issue, nonetheless, for HKTs, as they are commonly installed in water channels with a constant flow direction. Several HKTs companies are already exploiting this concept, such as HE-Powergreen [13].

Building upon the reviewed literature and a previous study presented by the authors in Ref. [14], the present work extends the findings by analyzing also a corotating pair of turbines, as well as arrays that consist of three turbines organized in different configurations. The use of an odd number of rotors is something quite unusual to date in both academic research and industrial prototypes, but it could be beneficial in all cases in which, for example, channels are too narrow for two pairs of rotors. The two-dimensional CFD model, selected upon previous experience for its good compromise between accuracy and computational cost, was first validated against experimental measurements provided in Ref. [15]. Then, the numerical model was used to carry out an extended analysis on the power augmentation mechanism in turbine arrays having different configurations. Special focus was given to loads by assessing instantaneous blade forces and generated torque. These loads were extensively studied by integrating data from locally sampled instantaneous kinematic data for each blade which facilitated the computation of dynamic polars, contributing to a more comprehensive understanding of the physical mechanisms within these compact setups. In addition, the wake of the different arrays was analyzed through the velocity deficit along various lateral distances and longitudinal velocity profiles. This analysis led to the definition of guidelines for designing an array of Darrieus turbines by identifying an efficiency improvement pattern based on two factors: i) the direction of rotation in relation to adjacent turbines, i.e., counter-rotating or corotating, and ii) the mode of mutual interaction between neighboring turbines, i.e., leeward or windward (see Fig. 1).

The study is structured as follows: The case studies illustrate the geometrical and operational aspects of the simulated array configurations. In the next section, the numerical methods are briefly outlined, covering the CFD domain, grid setup, the line average technique for the kinematic data sampling. The following section includes the validation of the numerical model and presents findings on instantaneous and average loads alongside corresponding kinematic data, in addition to the velocity profiles in the wakes of the arrays, followed by a discussion highlighting the significance of these results in the current state-of-the-art and industrial advancements. Finally, the last section summarizes the conclusions and outlines potential elements for future research.

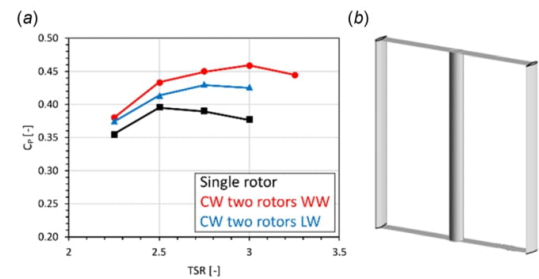


Fig. 1 (a) Performance curves obtained single and CW twin rotors [8] and (b) 3D scheme of the rotor adopted in the current study

Table 1 Characteristics of the turbine(s) model

Airfoil shape	DU06W200 (reversed)
Number of blades (#)	2
Diameter (m)	2
Chord (m)	0.25
Solidity (nc/D)	0.25
Tip speed ratio (TSR)	3
Freestream velocity (m/s)	2
Freestream turbulent intensity (%)	1

Case Study

The turbine used in this study is an industrial medium solidity rotor concept by HE-Powergreen, previously utilized in other CFD studies [16–18]. In hydrokinetic applications, low-medium solidity rotors reduce hydrodynamic loads, unlike high-solidity rotors necessary in wind applications to address the self-starting issue, which is not a concern in HKTs [18]. The selection of this specific rotor is justified as it operates, albeit in its final industrial design, within a multiple rotor setup in the Biffis canal in Trento, Italy. Hence, the findings from this analysis could directly benefit future practical applications. Table 1 presents the geometrical and operational characteristics of the rotor under investigation, while the performance curves for the adopted single and counter-rotating twin rotors is reported in Fig. 1(a), which was obtained via CFD in a recent study by the authors [8]. Figure 1(b) illustrates a schematic three-dimensional representation of the adopted rotor.

In this study, the performance of the turbine is evaluated both as a stand-alone unit and in multirotor array configurations consisting of either two or three turbines. The various layouts under investigation are summarized in Table 2, where each configuration is identified by an acronym that has been conceived to let the reader have at hand all the relevant info on the test case. The acronym is structured as follows:

- The number of turbines in the array is first indicated by a numeral (2 or 3);
- Similar to the numbering of airplane seats, each turbine is assigned a letter (A, B, or C) as shown in Fig. 2(c);
- The rotational orientation relative to neighboring turbines is denoted as either counter-rotating ("−") or corotating ("+"), as depicted in Fig. 2(a);
- The mutual interaction region is categorized as leeward ("l") or windward ("w"), as shown in Fig. 2(b).

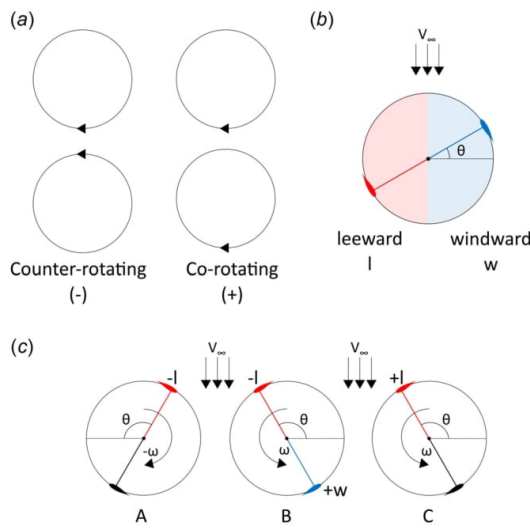
For example, Fig. 2(c) illustrates the schematic for case f, where the three turbines, from left to right, are represented as: 3(A | −l), 3(−l | B | +w), and (+l | C).

Methodology

Computational Fluid Dynamics Model. ANSYS FLUENT, version R20.2, commercial CFD software was used for all simulations,

Table 2 Study cases for the present study

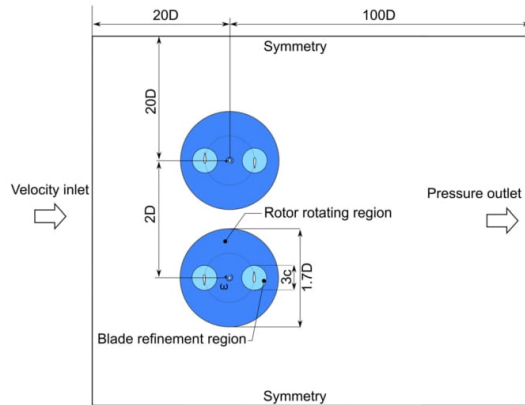
Case	No. of rotors	Acronym per rotor
a	1	SR
b	2	2(A -l) 2(-l B)
c	2	2(A -w) 2(-w B)
d	2	2(A +w) 2(+l B)
e	3	3(A -l) 3(-l B -w) 3(-w C)
f	3	3(A -l) 3(-l B +w) 3(+l C)
g	3	3(A +w) 3(+l B -w) 3(-w C)

**Fig. 2 (a) Counter-rotating versus corotating, (b) windward versus leeward, and (c) schematic representation of case f**

employing a two-dimensional, pressure-based unsteady RANS approach.

The authors relied on their previous experience with similar test cases [19,20] to make the selection of the numerical schemes and turbulence models. The temporal and spatial discretization levels were chosen based on comprehensive sensitivity studies that the authors conducted, making sure to comply with the dimensionless vorticity and Courant number thresholds recommended in Ref. [21]. To minimize numerical diffusion, the convection/diffusion terms were approximated using the second-order upwind convection scheme. For all simulations, an angular time-step of 0.4 deg was utilized in conjunction with the second-order implicit time advancing scheme. Simulation convergence was evaluated by measuring the average torque over one complete rotor revolution, setting as threshold a 0.1% variation between two consecutive cycles, which corresponds in this study to a simulation duration of approximately 25 revolutions, depending on the operating point. Turbulence closure was accomplished using the $k-\omega$ SST turbulence model [22].

The size of the computational domain was derived from previous research on a single rotor [19] and adjusted to account for the presence of multiple rotors and the associated blockage effect. The domain was enlarged by a distance S in the crosswise direction

**Fig. 3 Computational domain**

between the rotors, where S is taken as $2D$ in all cases as proven previously the optimum separating distance for this rotor [8]. In all cases, the computational domain included rotating zones for the rotors and a stationary zone for the wake flow field, separated by a nonconformal mesh utilizing the sliding mesh technique. The rotating domain was sized to be $1.7D$. A schematic representation of the domain dimensions and boundary conditions is shown in Fig. 3 for the two rotors case as an example.

A nonstructured mesh was utilized, resembling the one used in prior research [8,19,23]. Figure 4 shows the mesh adopted for one of the two rotors cases as an example.

The mesh is mainly made of triangular elements, which were most refined in the zones surrounding the airfoils (as shown in Fig. 4(e)). The closest cell layer from the wall was meshed with quad elements, comprising 50 inflation layers on the blades. These layers were sized to achieve a dimensionless wall distance of $y^+ < 1$, in line with the specifications of the $k-\omega$ SST turbulence model [22]. The rotating domains were divided into sections to ensure a gradual progression from fine elements adjacent to the airfoil to the coarse elements in the stationary domain. Two wake zones were created in the stationary domain, where the mesh density increased gradually from the rotors to the far field. The corresponding details are reported in Table 3.

Angle of Attack Sampling. In general, Darrieus turbines are unique in the fact that their blades rotate in a cycloidal manner, causing inherent variations in the aero/hydrodynamics along the azimuthal path. Complex flow phenomena involved in this functioning principle include blade-wake interactions, vortex shedding, and dynamic stall [24] (Fig. 5). Moreover, rotating blades cause a blockage effect that leads to the so-called “streamtube expansion,” which deflects the inflow that approaches the turbine. This phenomenon is more relevant in closely spaced turbines, in which flow deflection is the main cause of efficiency improvement or degradation [8]. As a result, simplified aerodynamic analyses are insufficient to accurately estimate the azimuthal variation of kinematic quantities such as the angle of attack and altered inflow. In the present study, to sample the local velocity field, the *LineAverage* (*LineAvg*) method [25] was employed (see Fig. 5), as prescribed by [26,27]. These studies demonstrated that the results obtained with the *LineAvg* method may have some discontinuities at low TSRs due to the interaction between separation vortices and the sampling line downstream of the airfoil. However, this is not a concern in the present case as selected operating points are within the stable functioning range. The *LineAvg* technique samples the undisturbed inflow velocity by computing an integral average along a closed line encircling the airfoil’s aerodynamic center, approximately located at 25% of the chord length. By doing so, the induced

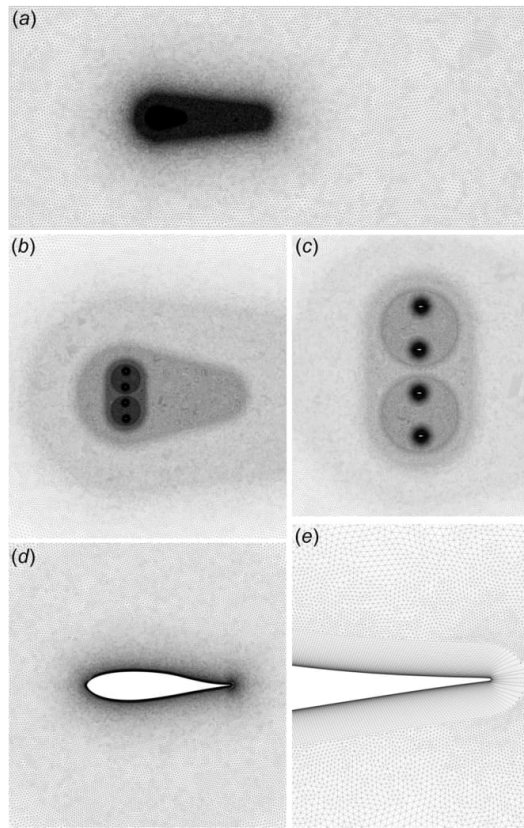


Fig. 4 Mesh details: (a) far field, (b) near wake, (c) rotating zones, (d) blade, and (e) trailing edge

Table 3 Mesh characteristics

Parameter	Value
Approximate # elements in stationary domain	500 K
# elements in each rotating domain	490,136
# nodes on the blade	1440
# inflation layers on the airfoil	60

velocity components are smoothed out during the averaging process, eliminating the effects of the profile bound circulation. After sampling the local velocities in the x- and y-directions, they are combined to determine the inflow deviation angle, ϕ and the local inflow velocity, $V_{\infty, \text{local}}$, which in turn is used with the rotor speed, ωR to compute both the angle of attack, α and the relative velocity, V_R . In this study, a circular sampling line with a radius $r = 1c$ and 80 evenly distributed sampling points was used.

Results and Discussion

In this section, a validation of the adopted CFD model is first carried out. Then, the performance of the different array configurations is compared against the stand-alone turbine in terms of cyclic average power coefficient, instantaneous torque coefficient, blade local kinematics, and wake behavior. In the graphs, the symbol [-] denotes non-dimensional parameters.

Computational Fluid Dynamics Model Validation. Several examples in the literature (e.g., [8] or [19]) have shown that

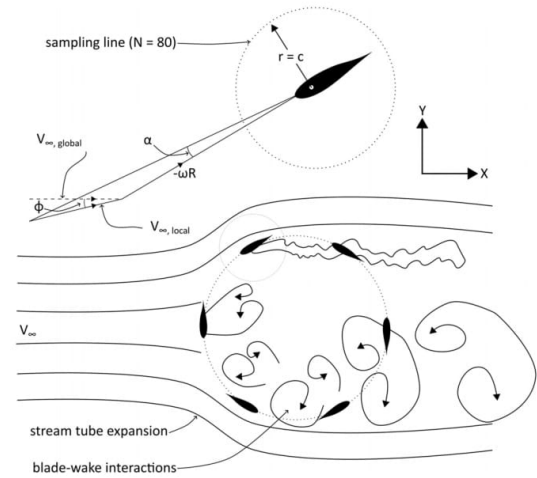


Fig. 5 Flow around a Darrieus rotor and the Linavg AoA sampling method

two-dimensional CFD simulations can be used to reliably analyze the physical mechanisms taking place past Darrieus turbine's blades, while they usually fall short when the actual turbine performance needs to be assessed, as tip losses and other 3D effects are neglected. In the present study, however, thanks to the presence of unique experimental measurements (presented in Ref. [15]), the CFD model was further validated. In particular, Fig. 6(a) reports a comparison between the results of the CFD model and experimental measurements in terms of power coefficient normalized over the maximum experimental power coefficient ($C_p/C_{p, \text{EXP, MAX}}$) at different tip-speed ratios for turbine B in a four-turbine array with the conventions illustrated in Fig. 6(b).

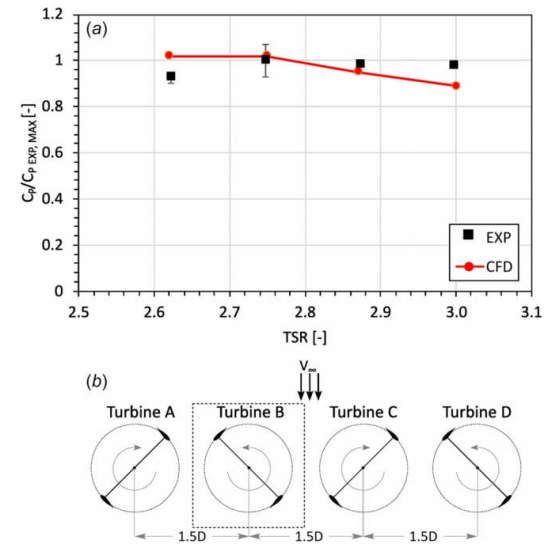


Fig. 6 (a) Comparison of experimental measurements with the 2D CFD model for normalized power coefficient at various tip-speed ratios and (b) schematic of the four-turbine array used in the validation

Despite the lack of tip losses' modeling, the CFD model generally shows close agreement with experimental data, showing as expected the largest discrepancies at the maximum and minimum tip-speed ratios, respectively. These deviations can be attributed to several factors, including: (1) the specific configuration of the experimental measurements that includes significant three-dimensional effects due to the relatively low aspect ratio of the turbine and (2) the presence of free surface effects, which were only partially corrected in experimental data based on the upstream water rise observed at each tip-speed ratio. Despite these discrepancies, the CFD model demonstrates good overall performance and effectively captures the general trend in power coefficient, consistent with the literature [19], thus demonstrating the capability of 2D CFD of modeling this type of complex test case. Moreover, it should be noted that the present study will mostly be carried out in a comparative way, i.e., analyzing the differences between different array configurations, thus practically purging the results from the common impact of three-dimensional effects.

Performance. Figure 7 shows the average power coefficient for the single rotor at TSRs 2.5 and 3, alongside that of each turbine within arrays, while also indicating the average power coefficient for the entire array. It is noteworthy that the optimal tip-speed ratio for the arrays is higher than that of a single rotor (see Fig. 1(a)). In particular, array simulations were conducted at TSR = 3, which is considered the optimal for the pair of rotors, whereas the optimum TSR for the single rotor is 2.5.

Regarding the two counter-rotating pairs (cases b and c), since each rotor within the array shares the same rotational scheme and mode of interaction with its neighboring rotor, their resulting performance is identical, with the leeward mode of interaction being the most efficient. On the other hand, the corotating pair of (case d) resulted in an asymmetrical performance, with an average power coefficient lower than that of the counter-rotating leeward pair (case b), and higher than that of the counter-rotating windward pair (case c). Interestingly, in this pair, and differently from the

counter-rotating pairs, the windward rotor resulted in a higher efficiency improvement than that of the leeward rotor. In the cases of three-rotor arrays, selections were made as extensions of the two-rotor scenarios (see Table 1): case e represents a blend of cases b and c, while case f combines cases b and d, and case g pairs cases c and d. The observed enhancement in the power coefficient aligns with the trends seen in the two-rotor setups. Notably, the combination of cases b and d exhibits the highest enhancement, whereas the combination of cases c and d shows the lowest improvement in efficiency.

Loads. Figure 8 illustrates the instantaneous torque coefficient of individual rotors categorized by their mode of mutual interaction. The variation in torque coefficient consistently aligns with the interaction mode, regardless of rotor count or rotational scheme. When considering windward interactions, most rotors mirror the torque of standalone turbines. However, within the range $90 \text{ deg} < \theta < 180 \text{ deg}$, rotors within arrays exhibit higher torque peaks. Specifically, $3(A \mid -l)$ performs best while $2(+l \mid B)$ shows comparatively lower efficiency. On the other hand, leeward interactions boost torque within the range of $0 \text{ deg} < \theta < 90 \text{ deg}$. Among these, $3(-w \mid C)$ is the most efficient, while $2(A \mid -w)$ is the least efficient. Turbines experiencing both leeward and windward modes of interaction within three-turbine arrays show enhanced torque across the entire upstream half ($0 \text{ deg} < \theta < 180 \text{ deg}$).

Notably, $3(-l \mid B \mid +w)$ demonstrates higher efficiency compared to $3(-l \mid B \mid -w)$ and $3(+l \mid B \mid -w)$ despite sharing similar interaction modes on each side but differing in rotational schemes.

The latter observation is crucial in understanding the power augmentation mechanism within three-rotor arrays, when framed in relation to findings from two-rotor arrays. To elaborate, insights from the two-rotor arrays reveal an efficiency hierarchy based on rotational orientation scheme and mode of mutual interaction: $-l > +w > -w > +l$. Following this sequence, constructing the most efficient three-rotor array necessitates a counter-rotating

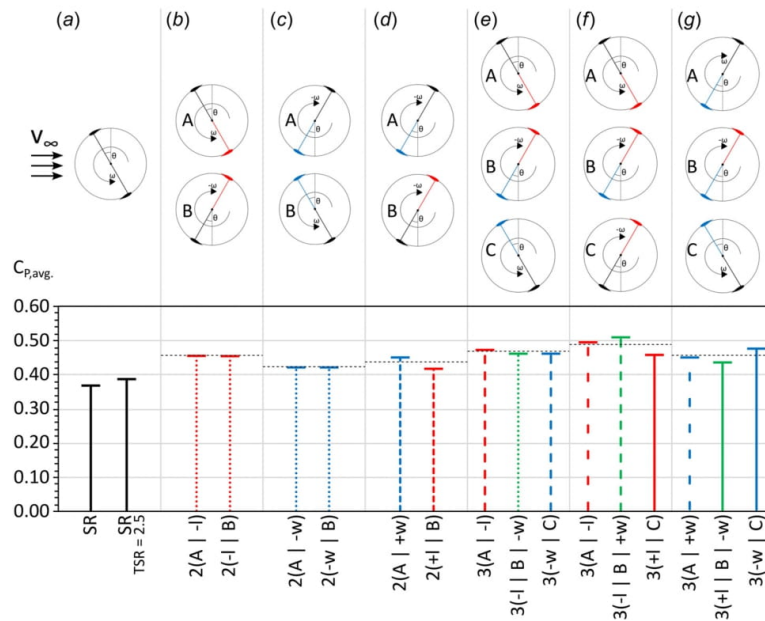


Fig. 7 Schematic representation of the simulated arrays and their resulting average power coefficient for each individual rotor and for overall cluster

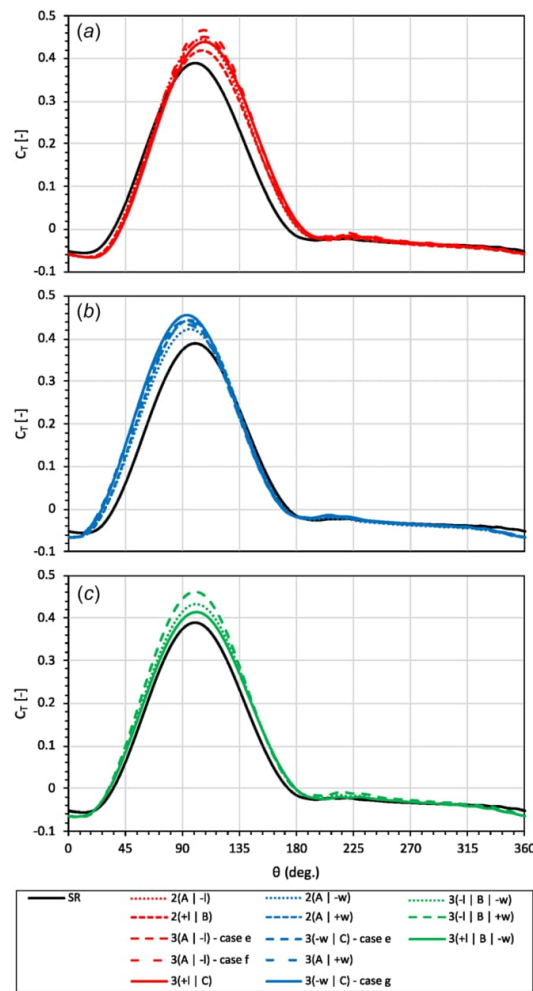


Fig. 8 Results of the instantaneous torque coefficient for each rotor individually (a) leeward, (b) windward, and (c) combination of leeward and windward

scheme on the leeward side for the middle rotor, paired with a corotating scheme on the windward side.

Kinematics. To get an insight into the physical mechanism driving the power augmentation of rotors within arrays in comparison to stand-alone turbines, the flow field is shown and the instantaneous kinematic data local to the blade is computed following the methodology outlined previously.

Figure 9 illustrates dimensionless velocity contours and streamlines, focusing on the flow field near the rotors across three different cases: a, b, and e. These three example cases demonstrate clearly how the inflow direction varies due to interactions between adjacent rotors. In case a, the flow around a single rotor exhibits a noticeable deviation on both leeward and windward sides, primarily due to streamtube expansion, showing also minimal lateral acceleration. Conversely, cases with arrays show significant acceleration on turbine sides, even those without mutual interaction. In case b, a one-sided interaction alters the leeward inflow, affecting the blade angle

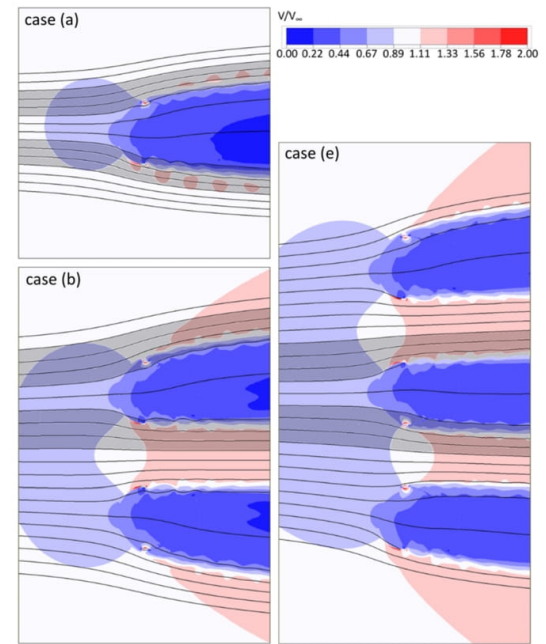


Fig. 9 Dimensionless velocity contours and streamlines for cases a (no interactions), b (one side interaction) and e (two sides interaction)

of attack. In case e, positioned within a three-turbine array, the central rotor exhibits two-sided interaction, leading to inflow blockage on both sides, fully suppressing the stream expansion effect.

To quantitatively emphasize the latter effect, Fig. 10 reports the variations in the instantaneous angle of attack, AoA and inflow deviation angle, ϕ based on mutual interaction modes. Like torque trends, cases exhibiting the same interaction mode showcase analogous patterns in AoA and ϕ variations. The most efficient turbine cases demonstrate the highest AoA peaks. It is evident that mutual flow interactions between adjacent rotors reduce inflow deviation angles, thereby increasing the angle of attack and thus torque generation. This phenomenon occurs only when the turbines operate at a sufficiently high tip-speed ratio, ensuring that no flow separation is expected during the AoA increase. Notably, while mutual interactions decrease ϕ in both leeward and windward halves compared to the stand-alone turbine, a notable torque increase occurs solely in the upstream part of these mutual interaction halves.

The results imply an increase in AoA and suggest a lag between stand-alone rotors and others. In leeward cases, turbines within arrays lag behind the stand-alone turbine, whereas in windward cases, the stand-alone turbine lags behind the other rotors. Cases involving both leeward and windward modes of interaction exhibit dual lags depending on the azimuth position: between 0 deg and 90 deg, the stand-alone rotor lags, while between 90 deg and 180 deg, the rotors within arrays lag, resulting in overall curves that overlay the curve of the stand-alone turbine. These curves depict higher AoA values across azimuth positions.

Figure 11 displays lift, drag, and moment coefficients for selected cases compared to the stand-alone turbine. Flow separation absence across cases results in lift hysteresis not dropping during pitch down but slightly rising due to dynamic effects. Blades within clusters exhibit similar lift hysteresis to the single rotor, differing in slope during pitch-up; all match during pitch-down. Drag coefficient

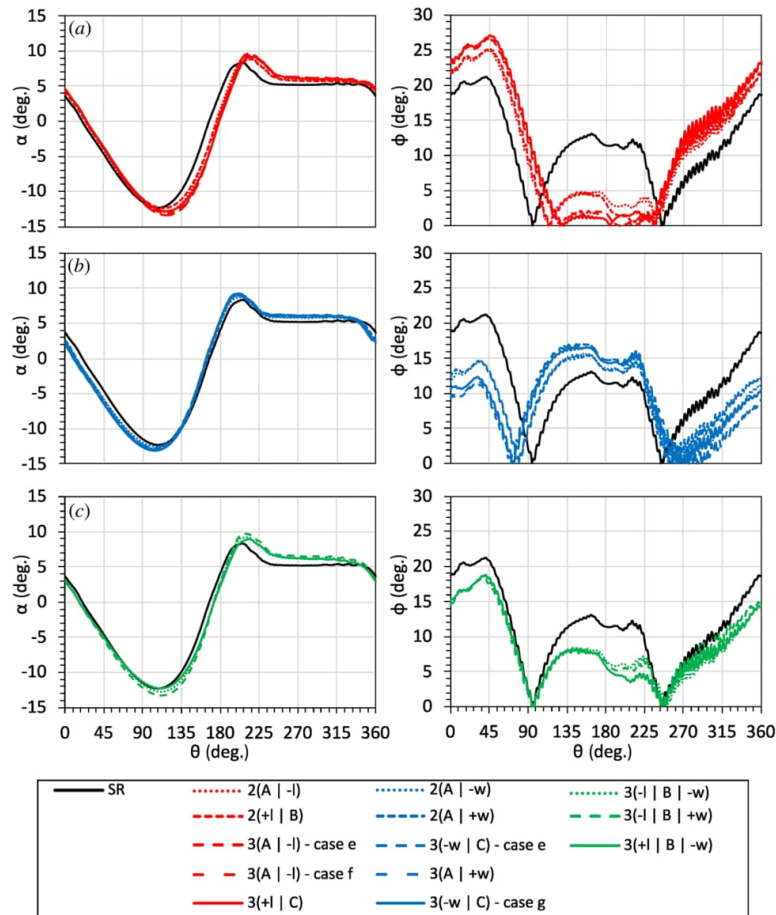


Fig. 10 Results of the instantaneous AoA for each rotor individually (a) leeward, (b) windward, and (c) combination of leeward and windward

trends are similar across cases, but differences occur when blades within clusters pitch-down which is attributed to dynamic effects. The pitching moment coefficient, on the other hand, shows different hysteresis behavior, with an almost constant pitching moment during pitch-up for the stand-alone turbine versus a gradual increase for all other cases. The extracted polars can be better understood if plotted against the azimuth position, θ as illustrated on the right-hand side of Fig. 9. In the leeward case, the lift coefficient notably increases within $90 \text{ deg} < \theta < 180 \text{ deg}$ compared to the stand-alone turbine.

Conversely, the windward case displays increased lift within $0 \text{ deg} < \theta < 90 \text{ deg}$, while the case that have both leeward and windward modes of interaction shows an overall lift increase across the entire upstream side $0 \text{ deg} < \theta < 180 \text{ deg}$. The observed lag in AoA variation is mirrored in the drag coefficient, exhibiting a slight peak increase in drag within array cases. The stand-alone turbine blade maintains a nearly constant pitching moment, whereas turbines within arrays depict a gradual increase throughout the entire upstream half.

Wake Analysis. To evaluate the wake behavior in each case and understand how the arrangement of the turbines within an array

could influence the wake recovery behavior, Fig. 12 displays the contours of dimensionless velocity for all assessed cases.

The wake of a stand-alone Darrieus turbine usually displays noticeable asymmetry (as in Fig. 12(a)), tilting toward the windward side because of the uneven load distribution across azimuthal positions [28]. However, at higher tip-speed ratios, this asymmetry tends to diminish, primarily due to the reduced variation in azimuthal loads and the absence of dynamic stall. Within the simplified context of a two-dimensional flow field that neglects three-dimensional vertical mixing, the wake of an isolated turbine gradually dissipates over distance, adopting a characteristic swirling and unsteady pattern resembling the von Kármán vortex streets. This pattern persists until the wake gradually regains the freestream velocity.

In the scenarios involving arrays of multiple counter-rotating turbines (cases b, c, and e), each interaction zone between adjacent turbines exhibits a symmetrical wake behavior. This pattern maintains an almost identical wake dissipation rate observed in a single stand-alone turbine (case a), thereby preserving its defining features. Conversely, arrays configured with a corotating scheme (cases d, f, and g) produce an asymmetrical wake pattern. This asymmetry triggers early wake mixing, resulting in a quicker wake recovery process.

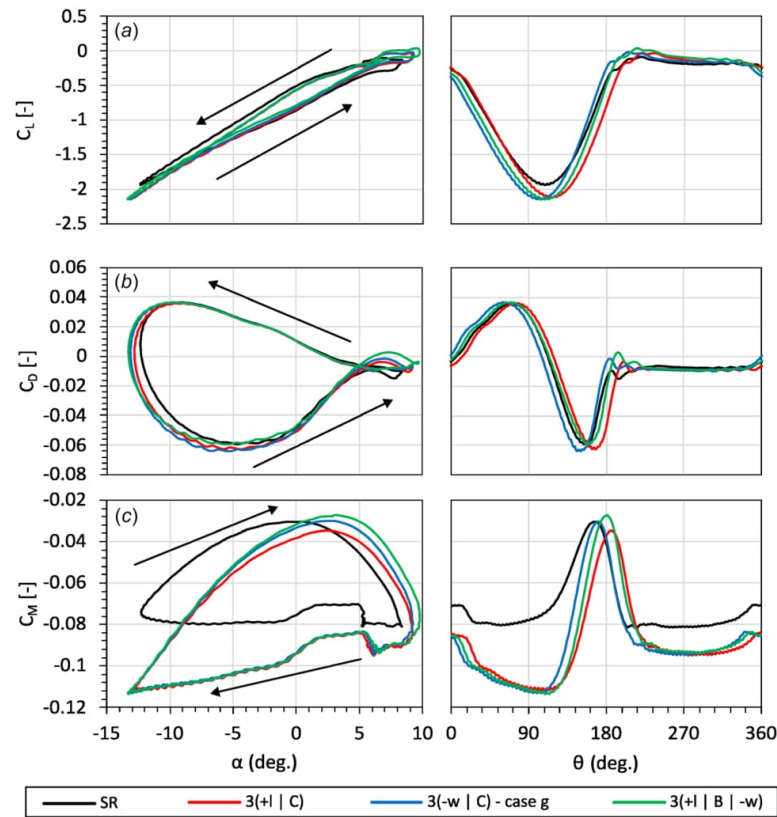


Fig. 11 Results of the extracted dynamic polars: coefficients of (a) lift, (b) drag, and (c) pitching moment

Figure 13 reports the lateral dimensionless velocity profiles along various longitudinal distances for a stand-alone turbine and two three-turbine array cases: one composed entirely of counter-rotating turbines (cases e) and the other consists of a mix of counter and corotating turbines (case f). At $X/D = 2$, there is a consistent velocity deficit across all cases, closely resembling that of the stand-alone turbine, accompanied by accelerated flow at the turbine sides. Minimal deflection is evident across the array cases at this point. However, at $X/D = 6$, clearer deflections in velocity profiles emerge, showcasing significant variations in velocity deficit influenced by the rotational direction of each rotor in cases featuring arrays.

The acceleration zones on the sides for the array cases persist similarly to $X/D = 2$. Moving to $X/D = 10$, notable changes occur, especially in case f, where profiles become highly asymmetrical with significant wake recovery, in contrast to case e, maintaining symmetry between the turbines at the sides and exhibiting increased velocity deficit. As we reach $X/D = 14$, all velocity profiles exhibit at least a 50% recovery from the minimum inflow velocity.

Discussion. The findings presented in this study suggest that a pair of Darrieus turbines, counter-rotating on the leeward side, is likely to produce the highest power augmentation. Following this, the corotating configuration is expected to exhibit moderate efficiency enhancement, while the counter-rotating windward setup shows comparatively lower efficiency improvement compared to a stand-alone rotor. This ranking aligns with recent conclusions drawn by Hara et al. [29] for VAWTs pairs and more recently by Sahebzadeh et al. [30] for a turbine pair experiencing skewed inflow

conditions. However, in composing an array comprising more than two turbines, maintaining the counter-rotating leeward configuration for all turbines is impossible. Specifically, a three-turbine array consisting solely of counter-rotating turbines would lead to two mutual interaction zones, inherently necessitating one zone on the windward side. This arrangement would not constitute an optimal design from an efficiency standpoint. However, in channels not large enough to host four rotors, it is important to note that this configuration still provides a non-negligible performance improvement with respect to an equivalent number of single rotors.

It is important to note that the findings of this study do not consider the three-dimensional nature of the flow. In a three-dimensional domain, power augmentation is expected to be lower due to vertical mixing. Nazari et al. [31] highlight that the three-dimensional flow nature induces faster wake recovery compared to the assumed two-dimensional flow, attributed to enhanced vertical mixing, especially for turbines with low aspect ratio. However, the present results, indicating faster wake recovery in cases of corotating turbines, are consistent with experimental observations by Müller et al. [5], at which they identified asymmetry within the wake zone of the corotating pair, demonstrating notable diffusion in both lateral and vertical directions compared to the counter-rotating pairs and stand-alone turbine scenarios.

According to present findings, optimizing arrays with more than two turbines involves maximizing zones with leeward modes of interaction. This strategy should encompass a corotating scheme, as it not only promises a higher overall efficiency boost for the array but also ensures quicker wake recovery.

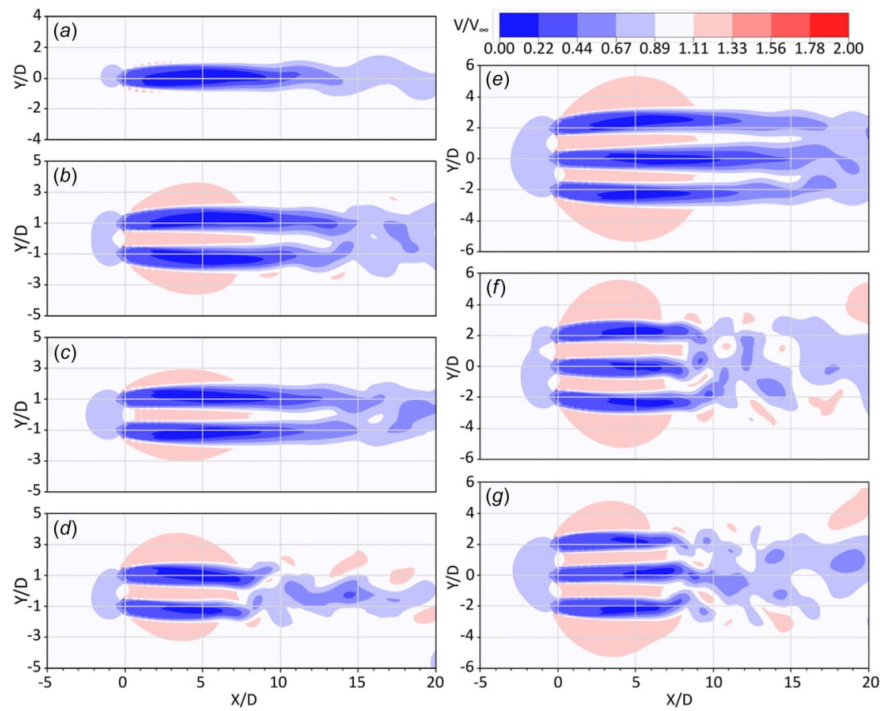


Fig. 12 Contours of dimensionless velocity for all evaluated cases and longitudinal velocity profiles

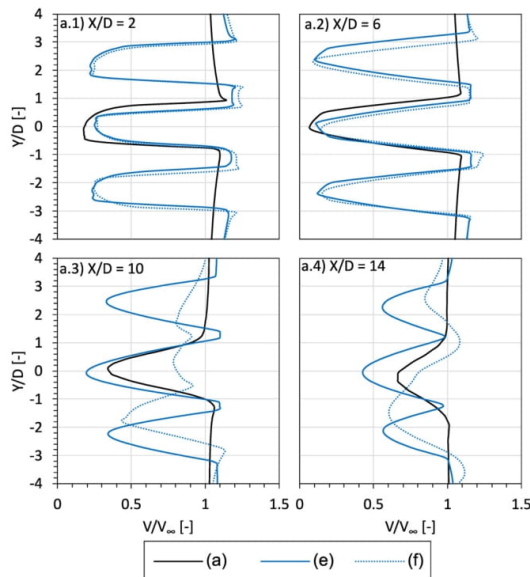


Fig. 13 Lateral velocity profiles at different longitudinal distances for cases a, e, and f

Conclusions

Several two-dimensional CFD simulations were conducted to assess various configurations of Darrieus turbine arrays comprising two or three turbines placed in close proximity. Each turbine was

classified based on the array size, rotational orientation, and the mode of mutual interaction with neighboring turbine(s). The conclusions drawn from these results are as follows:

- In two-turbine arrays, the efficiency improvement ranked in the following order across the three possible cases: Counter-rotating Leeward > Co-rotating > Counter-rotating Windward.
- Extending to three-turbine arrays, based on the conclusions drawn from the two-turbine array, the maximum efficiency improvement followed this order: Counter-rotating Leeward + Co-rotating > Counter-rotating Leeward + Counter-rotating Windward > Counter-rotating Windward + Co-rotating.
- The efficiency enhancement in closely deployed Darrieus turbines primarily results from minimizing inflow deflection within the areas of flow interaction.
- Reducing inflow deflection increases the angle of attack, thereby increasing lift force and, as long as the flow remains attached (at sufficiently high tip speed ratios), improving overall efficiency.
- Arrays incorporating at least one co-rotating scheme exhibit highly asymmetrical wakes characterized by rapid flow mixing, consequently enhancing wake recovery.

Data Availability Statement

The datasets generated and supporting the findings of this article are obtainable from the corresponding author upon reasonable request.

Nomenclature

AoA = angle of attack (deg)
 c = chord length (m)

CD = blade drag coefficient
 CL = blade lift coefficient
 CM = blade pitching moment coefficient
 CP = turbine power coefficient
 CT = turbine torque coefficient
 D = turbine diameter (m)
 R = turbine radius (m)
 Rec = chord-based Reynolds number
 V_∞ = freestream velocity (ms^{-1})

Greek Symbols

ϕ = inflow deviation angle (deg)
 θ = blade azimuth angle (deg)
 ω = rotor rotational speed, rad s^{-1}

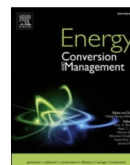
References

- [1] Fadil, J., Soedibyo., and Ashari, M., 2017, "Performance Comparison of Vertical Axis and Horizontal Axis Wind Turbines to Get Optimum Power Output," 15th International Conference on Quality in Research (QIR) International Symposium on Electrical and Computer Engineering, July 24–27, Bali, Indonesia, pp. 429–433.
- [2] Dabiri, J. O., 2011, "Potential Order-of-Magnitude Enhancement of Wind Farm Power Density Via Counter-Rotating Vertical-Axis Wind Turbine Arrays," *J. Renewable Sustainable Energy*, 3(4), p. 043104.
- [3] Vergaerde, A., De Troyer, T., Standaert, L., Kluczevska-Bordier, J., Pitance, D., Immas, A., Silvert, F., and Runacres, M. C., 2020, "Experimental Validation of the Power Enhancement of a Pair of Vertical-Axis Wind Turbines," *Renewable Energy*, 146, pp. 181–187.
- [4] Li, S., Li, Y., Yang, C., Wang, Q., Zhao, B., Li, D., Zhao, R., Ren, T., Zheng, X., Gao, Z., and Xu, W., 2021, "Experimental Investigation of Solidity and Other Characteristics on Dual Vertical Axis Wind Turbines in an Urban Environment," *Energy Convers. Manage.*, 229, p. 113689.
- [5] Müller, S., Muhawenimana, V., Wilson, C., and Ouro, P., 2021, "Experimental Investigation of the Wake Characteristics Behind Twin Vertical Axis Turbines," *Energy Convers. Manage.*, 247, p. 114768.
- [6] Zanforlin, S., and Nishino, T., 2016, "Fluid Dynamic Mechanisms of Enhanced Power Generation by Closely Spaced Vertical Axis Wind Turbines," *Renewable Energy*, 99, pp. 1213–1226.
- [7] Alexander, A. S., and Santhanakrishnan, A., 2020, "Mechanisms of Power Augmentation in Two Side-by-Side Vertical Axis Wind Turbines," *Renewable Energy*, 148, pp. 600–610.
- [8] Mohamed, O. S., Melani, P. F., Balduzzi, F., Ferrara, G., and Bianchini, A., 2023, "An Insight on the Physical Mechanisms Responsible for Power Augmentation in a Pair of Counter-Rotating Darrieus Turbines," *Energy Convers. Manage.*, 284, p. 116991.
- [9] Mohamed, O. S., Ibrahim, A., and El Baz, A. M. R., 2019, "CFD Investigation of the Multiple Rotors Darrieus Type Turbine Performance," *ASME Paper No. GT2019-9149*.
- [10] Zheng, H.-D., Zheng, X. Y., and Zhao, S. X., 2020, "Arrangement of Clustered Straight-Bladed Wind Turbines," *Energy*, 200, p. 117563.
- [11] Zhao, P., Jiang, Y., Liu, S., Stoesser, T., Zou, L., and Wang, K., 2021, "Investigation of Fundamental Mechanism Leading to the Performance Improvement of Vertical Axis Wind Turbines by Deflector," *Energy Convers. Manage.*, 247, p. 114680.
- [12] Parneix, N., Fuchs, R., Immas, A., and Silvert, F., 2016, "Efficiency Improvement of Vertical-Axis Wind Turbines With Counter-Rotating Layout," *Proceedings of the WindEurope Annual Event*, Hamburg, Germany, Sept. 27–29, pp. 1–8.
- [13] Cacciali, L., Battisti, L., and Dell'Anna, S., 2023, "Multi-Array Design for Hydrokinetic Turbines in Hydropower Canals," *Energies*, 16(5), p. 2279.
- [14] Mohamed, O. S., Melani, P. F., Soraperra, G., Brighenti, A., Balduzzi, F., and Bianchini, A., 2024, "Design Guidelines for Arrays of Closely Spaced Darrieus Turbines," *ASME Paper No. GT2024-121978*.
- [15] Mohamed, O. S., Melani, P. F., Soraperra, G., Brighenti, A., Betti, V., Schippa, L., Guerrero, M., Balduzzi, F., Ferrara, G., and Bianchini, A., 2024, "Three-Dimensional CFD-ALM-VOF Modeling of Hydrokinetic Turbines in Realistic Open-Channel Conditions," *Ocean Eng.*, 313(2), p. 119411.
- [16] Melani, P. F., Balduzzi, F., Ferrara, G., and Bianchini, A., 2021, "Development of a Desmodromic Variable Pitch System for Hydrokinetic Turbines," *Energy Convers. Manage.*, 250, p. 114890.
- [17] Mohamed, O. S., Melani, P. F., Papi, F., Balduzzi, F., and Bianchini, A., 2023, "Is the Actuator Line Method Able to Reproduce the Interaction Between Closely-Spaced Darrieus Rotors? A Critical Assessment on Wind and Hydrokinetic Turbines," *Energy Convers. Manage.*, 293, p. 117473.
- [18] Balduzzi, F., Melani, P. F., Soraperra, G., Brighenti, A., Battisti, L., and Bianchini, A., ATI Italian Termotechnics Association 2021, "Some Design Guidelines to Adapt a Darrieus Vertical Axis Turbine for Use in Hydrokinetic Applications," *E3S Web Conf.*, 312, p. 08017.
- [19] Balduzzi, F., Bianchini, A., Maleci, R., Ferrara, G., and Ferrari, L., 2016, "Critical Issues in the CFD Simulation of Darrieus Wind Turbines," *Renewable Energy*, 85, pp. 419–435.
- [20] Campobasso, M. S., Yan, M., Bonfiglioli, A., Gigante, F. A., Ferrari, L., Balduzzi, F., and Bianchini, A., 2018, "Low-Speed Preconditioning for Strongly Coupled Integration of Reynolds-Averaged Navier–Stokes Equations and Two-Equation Turbulence Models," *Aerosp. Sci. Technol.*, 77, pp. 286–298.
- [21] Balduzzi, F., Bianchini, A., Ferrara, G., and Ferrari, L., 2016, "Dimensionless Numbers for the Assessment of Mesh and Timestep Requirements in CFD Simulations of Darrieus Wind Turbines," *Energy*, 97, pp. 246–261.
- [22] Menter, F. R., 1994, "Two-Equation Eddy-Viscosity Turbulence Models for Engineering Applications," *AIAA J.*, 32(8), pp. 1598–1605.
- [23] Ibrahim, A. A., Elbaz, A. M. R., Melani, P. F., Mohamed, O. S., and Bianchini, A., 2022, "Power Augmentation of Darrieus Wind Turbine Blades Using Trapped Vortex Cavity," *J. Wind Eng. Ind. Aerodyn.*, 223, p. 104949.
- [24] Wang, Y., Sun, X. J., Zhu, B., Zhang, H. J., and Huang, D. G., 2016, "Effect of Blade Vortex Interaction on Performance of Darrieus-Type Cross Flow Marine Current Turbine," *Renewable Energy*, 86, pp. 316–323.
- [25] Jost, E., Klein, L., Leipprand, H., Lutz, T., and Krämer, E., 2018, "Extracting the Angle of Attack on Rotor Blades From CFD Simulations," *Wind Energy*, 21(10), pp. 807–822.
- [26] Melani, P. F., Balduzzi, F., Ferrara, G., and Bianchini, A., 2020, "How to Extract the Angle Attack on Airfoils in Cycloidal Motion From a Flow Field Solved With Computational Fluid Dynamics? Development and Verification of a Robust Computational Procedure," *Energy Convers. Manage.*, 223, p. 113284.
- [27] Mohamed, O. S., Elbaz, A. M. R., and Bianchini, A., 2021, "A Better Insight on Physics Involved in the self-Starting of a Straight-Blade Darrieus Wind Turbine by Means of Two-Dimensional Computational Fluid Dynamics," *J. Wind Eng. Ind. Aerodyn.*, 218, p. 104793.
- [28] Khosravi, M., Sarkar, P., and Hu, H., 2016, "An Experimental Investigation on the Near Wake Characteristics of a Darrieus Vertical-Axis Wind Turbine," *AIAA Paper No. 2016-1732*.
- [29] Hara, Y., Jodai, Y., Okinaga, T., and Furukawa, M., 2021, "Numerical Analysis of the Dynamic Interaction Between Two Closely Spaced Vertical-Axis Wind Turbines," *Energies*, 14(8), p. 2286.
- [30] Sahebzadeh, S., Rezaeiha, A., and Montazeri, H., 2022, "Vertical-Axis Wind-Turbine Farm Design: Impact of Rotor Setting and Relative Arrangement on Aerodynamic Performance of Double Rotor Arrays," *Energy Rep.*, 8, pp. 5793–5819.
- [31] Nazari, S., Zamani, M., and Moshizi, S. A., 2018, "Comparison Between Two-Dimensional and Three-Dimensional Computational Fluid Dynamics Techniques for Two Straight-Bladed Vertical-Axis Wind Turbines in Inline Arrangement," *Wind Eng.*, 42(6), pp. 647–664.



Contents lists available at ScienceDirect

Energy Conversion and Management

journal homepage: www.elsevier.com/locate/enconman

An insight on the key factors influencing the accuracy of the actuator line method for use in vertical-axis turbines: Limitations and open challenges

Omar S. Mohamed, Pier Francesco Melani, Francesco Balduzzi, Giovanni Ferrara, Alessandro Bianchini*

Department of Industrial Engineering, Università degli Studi di Firenze, Via di Santa Marta, 3, 50139 Firenze, Italy

ARTICLE INFO

Keywords:

Actuator Line Model (ALM)
 Darrieus
 Vertical Axis Wind Turbine (VAWT)
 Hydrokinetic Turbines
 Computational Fluid Dynamics (CFD)
 Dynamic stall

ABSTRACT

Darrieus vertical-axis turbines are known for their complex aerodynamics connected to the continuous change in the angle of attack experienced by the blades, which often exceeds the static stall limit. Low fidelity tools such as the Blade Element Momentum Theory have been shown lately not to provide sufficient levels of accuracy, while the medium-fidelity Actuator Line Method (ALM) has been increasingly applied to Darrieus rotors. In this method, the blade-flow interaction is modeled as an equivalent momentum loss calculated introducing equivalent aerodynamic forces into the computed Computational Fluid Dynamics (CFD) domain. This strongly reduces the computational cost in comparison to blade-resolved CFD, allowing ALM to be used in three-dimensional problems, e.g., multiple rotors, floating offshore, etc. While several corrections and guidelines have been recently proposed to tailor ALM to Darrieus turbines, issues are still open on how to improve accuracy.

The present study aims at assessing to what extent the three main factors of the ALM theory, namely the quality of input polar, the dynamic stall modeling, and the force insertion in the domain, influence the overall accuracy of the method. In particular, this unprecedented understanding is enabled by the novel use of a “frozen ALM”, i.e., an ALM method fed by the aerodynamic forces calculated by blade-resolved CFD, which allowed to separate the contributions coming from airfoil performance analysis and force projection in the domain. Based on the results, three main important conclusions are drafted out: i) for high and medium tip-speed ratios, provided that the aerodynamic forces are correct, the ALM method is able to generate extremely accurate solutions of the flow field, almost equivalent to blade-resolved CFD; ii) the relevance of the kernel's shape and smearing function is largely overestimated and current knowledge is adequate for the model to be set; iii) a better dynamic stall model is indeed the real key factor that could lead to an improvement of the ALM accuracy.

1. Introduction

The performance of a Darrieus turbine is characterized by the instantaneous variation of the angle of attack (AoA), which can even exceed the static stall limit at some azimuthal positions, thus triggering the dynamic stall phenomenon [1]. This issue is more relevant at low and medium tip-speed ratios (TSRs) [2]. Nevertheless, some specific features associated with the Darrieus turbine functioning principle and architecture make it particularly attractive for some special applications, e.g., multiple-rotor arrays [3], Floating Offshore Wind Turbines (FOWTs) [4], urban wind power generation [5], and energy generation from small watercourses [6]. On the other hand, the intrinsic complexity of blade aerodynamics has the main drawback of making their simulation and design very complex. In particular, recent studies have pointed

out that engineering methods like the Blade Element Momentum (BEM) theory, although empowered by recent modifications that enhanced their accuracy, can hardly capture some of the involved physics, paving the way for a larger use of higher fidelity techniques, among which Computational Fluid Dynamics (CFD) plays a key role [7–9]. However, although using blade-resolved CFD gives an accurate solution to the Darrieus rotor flow field and load estimations, it is still computationally prohibitive for three-dimensional analyses due to strict requirements in terms of mesh and timestep accuracy [10]. To improve on design, a more reliable, but computationally-affordable, simulation tool is therefore needed. To this end, hybrid simulation methods are increasingly welcome. In particular, Sørensen and Shen [11] first introduced a few years ago the concept of the Actuator Line Method (ALM) for Horizontal Axis Wind Turbines (HAWTs) as a realistic alternative between the blade-resolved CFD and BEM. The ALM is based on inserting the lift and

* Corresponding author.

E-mail address: alessandro.bianchini@unifi.it (A. Bianchini).<https://doi.org/10.1016/j.enconman.2022.116249>

Received 9 July 2022; Received in revised form 12 September 2022; Accepted 14 September 2022

Available online 23 September 2022

0196-8904/© 2022 Elsevier Ltd. All rights reserved.

Nomenclature			
<i>Latin</i>		β	Gaussian kernel width [m]
A_M	Berg model tuning constant [-]	β_c	Gaussian kernel width in chord direction [m]
c	Chord length [m]	β_t	Gaussian kernel width in thickness direction [m]
C_D	Drag coefficient [-]	θ	Azimuth Angle [deg.]
C_L	Lift coefficient [-]	ω	Angular speed [rad/s]
C_N	Normal force coefficient [-]	<i>Abbreviations and Acronyms</i>	
C_T	Tangential force coefficient [-]	ALM	Actuator Line Method
D	Turbine diameter [m]	aniso	Anisotropic Gaussian
E^2	Mean square error [-]	AoA	Angle of attack [deg.]
F_b	Body force term [N]	BEM	Blade element momentum
F_N	Normal force [N]	BSC	Blade spoke connection
F_T	Tangential force [N]	CFD	Computational fluid dynamics
F_X	X-direction force [N]	CSS	Cubic spline smoothing
F_Y	Y-direction force [N]	FLOWT	Floating offshore wind turbine
h	Cell characteristic dimension [m]	GG	Anisotropic Gaussian-Gumbel
n	Number of blades [-]	HAWT	Horizontal axis wind turbine
u	X-velocity [m/s]	HKT	Hydrokinetic turbine
v	Y-velocity [m/s]	iso	Isotropic Gaussian
V_∞	Undisturbed wind velocity in axial direction [m/s]	LES	Large eddy simulation
V_{rel}	Relative wind velocity [m/s]	PIV	Particle image velocimetry
<i>Greek</i>		SST	Shear stress transport
ρ	Fluid density [kg/m ³]	TSR	Tip Speed Ratio
		URANS	Unsteady Reynolds-Averaged Navier-Stokes

drag forces into momentum equations as body forces distributed on a line that represents the blade. Thus, there is no need to resolve the blade boundary layer, which makes the ALM computationally far less expensive compared to the conventional blade-resolved CFD. The ALM has been extensively developed in the last few years for HAWTs to account for tip and root losses [12] and chord length variation [13]. Other works have validated the use of ALM coupled with Large Eddy Simulation (LES) in advanced applications such as analysis of near and far wakes [14] as well as the wind farm wake interaction with the atmospheric boundary layer [15].

Moving from these successful study cases, few studies also discussed the use of ALM for Darrieus rotors. Shamsoddin and Porte-Agel [16] applied the ALM to a Darrieus wind turbine for the first time to analyze the wakes using LES. Potential sources of discrepancies between simulated and measured data were attributed to the limitations in the numerical model, experimental errors in measurements, and inaccuracies in the tabulated airfoil data. Bachant et al. [17] found that RANS ALM overpredicts the power coefficient at high tip-speed ratios, ascribing the sources of error to either unmodelled bodies (e.g., drag due to spokes) or limitations in the aerodynamic data (e.g., the effect of virtual camber or dynamic stall). Mendoza et al. [18] compared the ALM predictions of a Darrieus turbine flow field with experimental data and attributed the source of mismatch to the oversized kernel width β , as well as to the sensitivity on the predicted aerodynamic forces. Zhao et al. [19] showed that RANS ALM overpredicts the thrust coefficient at almost all tip-speed ratios; this has been explained by the authors as a cause of the blockage effect in the solution domain, in addition to the lack of data regarding the turbulence intensity of experimental results. Mendoza and Goude [20] evaluated the ALM sensitivity to the aerodynamic coefficients by using two different sets of polar data, one measured experimentally [21] that showed underestimated power coefficient values, and the other exported from XFOIL [22] that showed an overestimation of the power coefficient at low tip-speed ratios. More recently, Melani et al. [23,24] proposed a number of submodels and corrections to tailor the ALM for use in Darrieus VAWTs, obtaining notable improvements in the overall accuracy, while still seeing some general overestimation of the performance, especially in the downwind region of the rotor.

Based on this background, one can say that the implementation of the ALM for Darrieus turbines still lacks a detailed investigation to determine the sources of discrepancy versus CFD, so as to understand the areas where improvements can be made. Generally speaking, three main issues are currently deemed to be responsible for existing lacks in modeling fidelity, i.e.: i) the quality of airfoil polar data and its ad hoc correction models [25,26], e.g., flow curvature effects; ii) the dynamic stall modeling, which is especially critical at low and medium tip-speed ratios [27]; iii) the insertion of the body forces into the domain, which is usually through the standard *isotropic two-dimensional Gaussian kernel* [13]. However, no consensus still exists on the relative impact of each of these effects.

Moreover, it should be noted that while the first and second issues are intrinsically related to the characteristics of Darrieus rotor aerodynamics, the third issue could be relevant also when applying the ALM to a HAWT. This is testified by the fact that the use of a more physically related Gaussian kernel has been suggested lately also for HAWTs. Martínez-Tossas et al. [28] investigated the influence of using a cylindrical anisotropic Gaussian kernel. They concluded that using optimal scales for the kernel widths allows the ALM to resolve small flow structures such as those found near the blade tip. By doing so, the ALM is capable of predicting the blade tip load without using tip loss correction. Cormier et al. [29] studied the effect of the ALM force smearing function, finding that the force projection function does not cause a significant difference in the tip vortex properties, and attributed the accuracy of reproducing the near-wake flow field to the mesh refinement. Churchfield et al. [30] tested an anisotropic Gaussian kernel, observing a more compact body forces distribution and a higher capability of the ALM to capture the tip vortices and their consequent effects more accurately. Martínez-Tossas et al. [31] used an analytical approach for a static airfoil to optimize the scales of the circular and elliptical kernel widths (β_t and β_c). The latter approach was based on comparing the potential flow solution over a Joukowski airfoil with that of an ALM solution that utilizes the aerodynamic coefficients exported from the former analytical solution, namely the *Frozen ALM*.

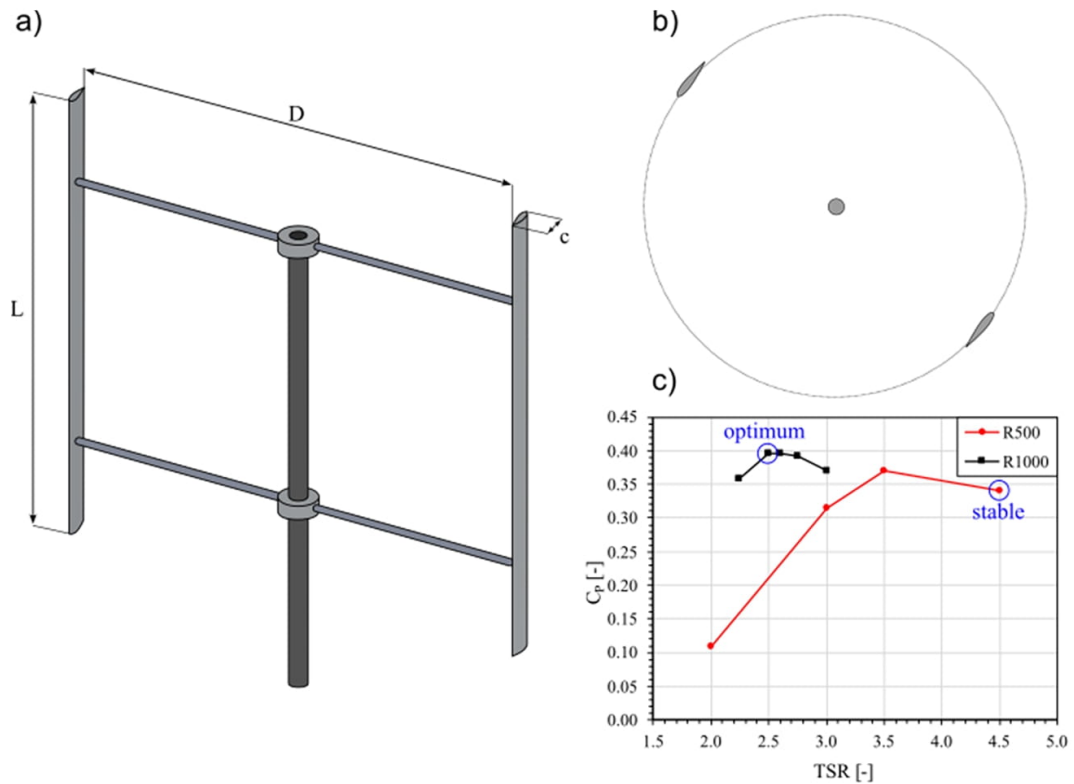


Fig. 1. a) generalized schematic representation of the Darrieus rotor, b) top view schematic of the Darrieus rotor, and c) performance curves of R500 and R1000.

1.1. Outline and contribution of the study

The present study is the outcome of a several-year research that the wind energy research group at the University of Florence has been carrying out over the last years focused at making computationally affordable those higher-fidelity methods that could provide the necessary step forward in Darrieus VAWT technology. More specifically, in [1] a novel method to extrapolate the angle of attack on a Darrieus turbine from a CFD-computed flow field. This passage was indeed of capital importance because it represented the only way to get useful benchmarking data from CFD. Thanks also to the information gathered with the new method, in [24] a number of corrections, sub-models and new methods have been presented that allowed tailoring the ALM to Darrieus VAWTs, while leaving some issues open on how to further improve the accuracy of the ALM.

The present study fills these open issues, providing an unprecedented overview on the specific research areas to be addressed to improve the accuracy of this method is highlighted, with particular focus on the actual influence of polar data quality, dynamic stall model, and force insertion function on the load estimation accuracy, as well as on the reproduction of the flow field.

This has been achieved by a novel method developed specifically for this study, which is based the application to Darrieus VAWTs of the *Frozen ALM* approach originally introduced by Martínez-Tossas et al. [31]. As will be shown in the paper, in the newly developed *Frozen ALM* the aerodynamic forces are determined from a blade-resolved CFD, thus enabling the elimination of the uncertainties related to the quality of the polar data or the dynamic stall model. By doing so, any possible discrepancy between the *Frozen ALM* and the blade-resolved CFD results must be attributed to the kernel function. The possibility of splitting

these contributions has never been possible before, thus always leaving doubts on the relative impact of each contribution and on their mutual influence.

Specifically, the objectives of this work are to: i) isolate and investigate the effect of the kernel shape and size; ii) highlight the real influence of the polar data and the dynamic stall modeling on the solution fidelity; iii) evaluate the solution sensitivity to the grid size for each of the evaluated kernel shapes.

Three kernel functions were investigated: isotropic Gaussian (*iso*) [13], anisotropic Gaussian (*aniso*) [32], and anisotropic Gaussian-Gumbel (*GG*) [33]. Unlike the cylindrically shaped isotropic function, the other two non-isotropic Gaussians make the shape of the projected body forces look more similar to the turbine blade. The latter non-isotropic functions have been previously applied to the ALM for HAWTs, but to the best of the authors' knowledge, they have not been studied in the context of Darrieus rotors to date.

The structure of this paper is as follows: In Section 2, details on the geometrical and operational characteristics of the selected test cases are demonstrated. Section 3 outlines the numerical methodology used in the present study, including a brief description of the ALM base formulation, *Frozen ALM* algorithm, and CFD set-up. The results are given in Section 4, where firstly a calibration of the kernel size is made using *Frozen ALM* for a static airfoil, followed by the *Frozen ALM* results for turbine cases. Thereafter, the significance of the kernel size is evaluated using the standard ALM. Then, grid sensitivity analysis is conducted for different kernel shapes. And finally, comparative analyses and remarks are highlighted. Section 5 presents a discussion of the results, emphasizing their relevance to the current state-of-the-art. The conclusions and future work are presented in Section 5.

Table 1

Details of test rotors selected for the investigation.

Dimension	R1000	R500
Number of blades [-]	2	2
Diameter, D [m]	2	1
Chord, c [m]	0.25	0.06
Rotor solidity (Nc/D), σ [-]	0.25	0.12
Airfoil	DU06W200	NACA0018
Blade-spoke connection [x/c]	0.25	0.25

Table 2

Test operating conditions.

Inlet value	R1000	R500
Tip-speed ratio [-]	2.5	4.5
Inflow fluid [-]	Water	Air
Inflow speed [m/s]	2	9.1
Inflow temperature [°C]	25	20
Turbulence intensity [%]	0.24	0.24

2. Study cases

To generalize the findings of this study for all Darrieus turbines, i.e. both wind and hydrokinetic ones, two different rotors have been considered, labelled as “R1000 HKT” and “R500 VAWT”, respectively. Both turbines are of H-Darrieus type. Fig. 1 shows a schematic representation of the H-type Darrieus rotor and the performance curves for both R1000 and R500 rotors. More details on the geometrical and operational characteristics are shown in Table 1 and 2, respectively.

- **R1000**: industrial two-blade Darrieus HKT, for which multiple design configurations have been investigated in a previous work [6]. It has been also studied in [35], where an investigation on a possible active pitch control strategy has been conducted using ALM. It is also worth remembering that for this rotor data from a high-fidelity, blade-resolved, two-dimensional CFD simulation are available; this dataset was used in the present study both for validation purposes and to obtain the input data needed for the Frozen ALM cases;
- **R500**: real two-blade, H-shaped Darrieus VAWT, tested in different studies [36,37] in the Open Jet Facility (OJF) at TU Delft. For this machine, both three-dimensional flow fields [36] and azimuthal blade force profiles [37] are available from *ad hoc* post-processing of measured PIV (Particle Image Velocimetry) data. To further support the validation process, two-dimensional blade-resolved CFD simulations from Melani et al. [38] are also here employed as a benchmark next to experimental measurements.

As apparent, the choice of these rotors for the investigation is

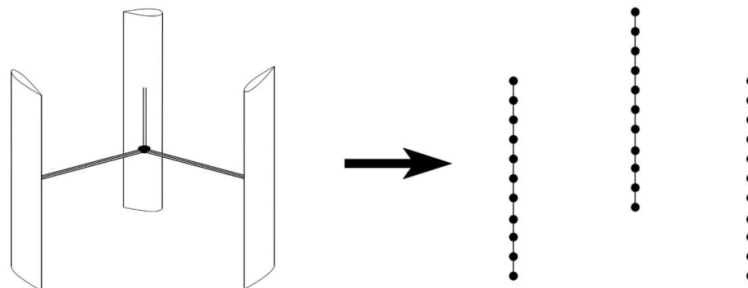


Fig. 2. Schematic representation of the ALM for straight-bladed Darrieus turbine.

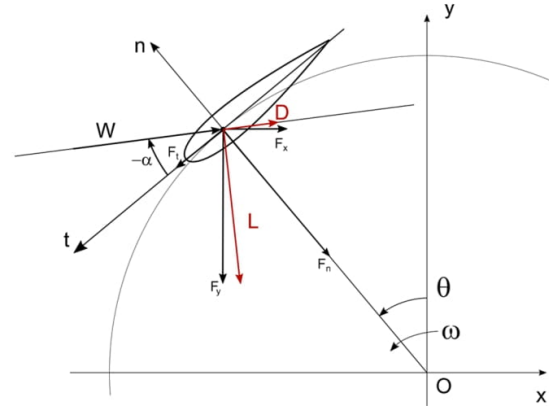


Fig. 3. Conventions adopted for the present study.

justified not only by the quantity and quality of the corresponding available benchmark data, but also by the possibility of generalizing the findings of this paper over a significant range of turbine architectures, solidities, and airfoils. Furthermore, both test cases fall in the low-to-medium solidity range, which is particularly indicated for hybrid simulation models, as their accuracy in fact strongly decreases going towards high-solidity configurations [39]. Furthermore, the choice of the operating conditions (see Table 2) was based on the will of assessing the ALM tool in different modes of operation. For the R1000, the optimum TSR of 2.5 was selected, since a significant impact of the dynamic stall model has been proven to exist. On the other hand, a TSR of 4.5 was selected for the R500, in which the operation is stable, and the dynamic stall phenomenon is not relevant.

3. Methods

3.1. Actuator line method

3.1.1. ALM algorithm

The ALM code used in the present study is an in-house code, tailored specifically for vertical-axis turbines and it has been validated already in previous works [23,34,35]. The ALM code is implemented through a custom User Defined Function (UDF) in ANSYS® FLUENT® (version 2020 R2).

In the ALM, lift and drag forces are projected into lines that represent the blades. Thus, no need to model the blade boundary layer. The aerodynamic forces are inserted into the computational domain of the CFD solver as a source term along the blade span. Fig. 2 shows a schematic representation of the ALM for a straight-bladed Darrieus turbine.

The modified governing equations: (1) continuity and (2)

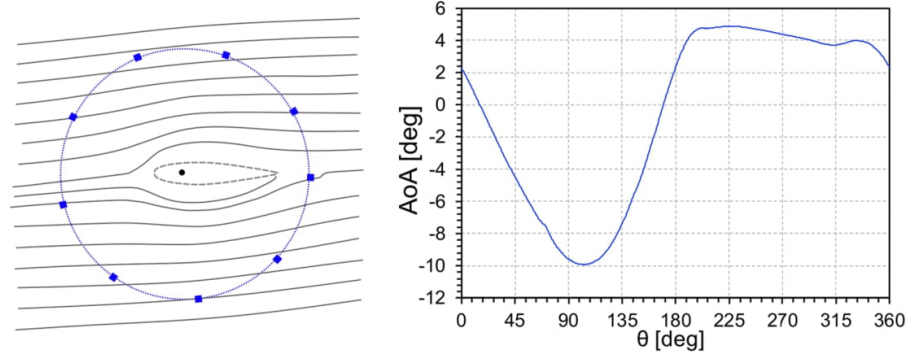


Fig. 4. Schematic comparison between the Tangent and the LineAverage methods.

momentum reads:

$$\nabla \cdot \vec{V} = 0 \quad (1)$$

$$\frac{D\vec{V}}{Dt} = -\frac{1}{\rho} \nabla p + \nu \nabla^2 \cdot \vec{V} + F_b \quad (2)$$

where $\frac{D\vec{V}}{Dt}$ represents the acceleration and the convection term, $-\frac{1}{\rho} \nabla p$ is the pressure gradient, $\nu \nabla^2 \cdot \vec{V}$ is the diffusion term, and F_b is the inserted body forces term. The body forces are computed using the local aerodynamic forces which read as:

$$F_L = \frac{1}{2} \rho V_{rel}^2 C_L \quad (3)$$

$$F_D = \frac{1}{2} \rho V_{rel}^2 C_D \quad (4)$$

where V_{rel} is the relative velocity, C_L and C_D are the coefficients of lift and drag, respectively. Normally, C_L and C_D are obtained using tabulated data based on the local angle of attack AoA, where AoA and W are determined from velocity sampling each time step, as shown in Fig. 3. To avoid non-physical discontinuities, the body forces have to be smeared out into the computational domain properly via a regularization kernel. Where $\eta(r)$ is the regularization kernel smearing function, the body forces f_b reads:

$$f_b = f_{2D} \otimes \eta(r) \quad (5)$$

The main features of the ALM code used in this study [24] are:

- **Velocity sampling:** The *LineAverage* method is used in this work to sample the local velocity through integral averaging of the undistributed inflow along a circular line around the hypothetical blade, this line has a radius $r = 1c$ and a resolution $N = 80$ of equally spaced points along the sampling line, as shown in Fig. 4. This method was originally proposed for HAWTs by Jost et al. [40] and has been verified and validated against other methods by the authors in recent works on VAWTs [1,2]. To minimize the background noise [34], the *LineAverage* method is coupled with low-pass filtering of the sampled data by using a dedicated Cubic Spline Smoothing (CSS) algorithm [41]. At the beginning of each rotor revolution, the latter ad hoc filtering strategy is applied to the AoA signals coming from the previous revolution, therefore, no real-time processing is needed. The smoothed AoA signal is then derived over time and used as an input for the dynamic stall model.
- **Polar data:** The quality of the polar data plays a vital role in the capability of the ALM code to predict the rotor performance accurately, hence, proper modeling of the flow field around the rotor. For small Darrieus rotors such as the two case studies considered in the

current work, the operating range of Reynolds numbers is relatively low. Thus, the influence of laminar separation bubbles on the aerodynamic performance of the airfoil is more relevant. For the R1000 rotor, the pre-stall aerodynamic coefficients were obtained via blade resolved CFD for $Re \in [7.5 \times 10^5, 2 \times 10^6]$. The 4-equations γ - $Re\theta$ transition model [42] was employed for the closure of the turbulence problem. On the other hand, for the R500 rotor, the pre-stall aerodynamic coefficients were calculated with the viscous panel method software Xfoil [22], since a benchmark against more complex methods proved them to be sufficiently robust. Both sets of data have been extended over the deep stall region, i.e. up to $AoA = \pm 180^\circ$, via the Viterna-Corrigan model [43], as recommended by Bianchini et al. [44] for VAWT applications.

- **Berg dynamic stall model:** To consider the unsteadiness effect on the blade loads, the dynamic stall model from Berg [45] has been used in the ALM code for the test case of R1000, employing the formulation proposed in [34]. In brief, the Berg model involves the dynamic effects by introducing a delay in the measured angle of attack that is proportional to the flow reduced frequency via a coefficient γ_i . It must be noted how the latter, which is given in the model as a semi-empirical function of the airfoil thickness and the freestream Mach number, was treated in this work as a free parameter. The airfoil used in the R1000 rotor, DU-06-W-200 profile, is in fact way thicker than those tested in the original work of Gormont [46] for the derivation of this correlation. Lift and drag coefficients are eventually obtained as a linear interpolation between the static and dynamic ones, weighted over the constant A_M . In the present work, A_M was set to be 6 and γ_i to 1.5 based on the dedicated calibrations that were conducted in [35] for the R1000 rotor. However, as will be discussed later on the study, one should bear in mind that a more effective dynamic stall model is indeed probably the real missing element in VAWT simulation and will deserve attention in the near future, either trying to tune more advanced ones like the Beddoes-Leishman model, or so synthesize new ones specifically tailored for these turbines.
- **Pitching moment and BSC corrections:** It must be noted how, in order to take the flow curvature phenomenon into account in the simulation campaign, the tabulated data for both R1000 and R500 rotors were derived for a virtually cambered airfoil, obtained by proper conformal mapping from the geometric one according to the rotor chord-to-radius ratio c/R , blade-spoke connection point (BSC) and TSR [47]. The latter was selected as the highest available in the simulation campaign. This procedure, already present in previous ALM tools from the authors [48,49] is supposed to be more accurate than the ones used in the existing software, which are mainly based on an algebraic correction of the sampled AoA [50,18].

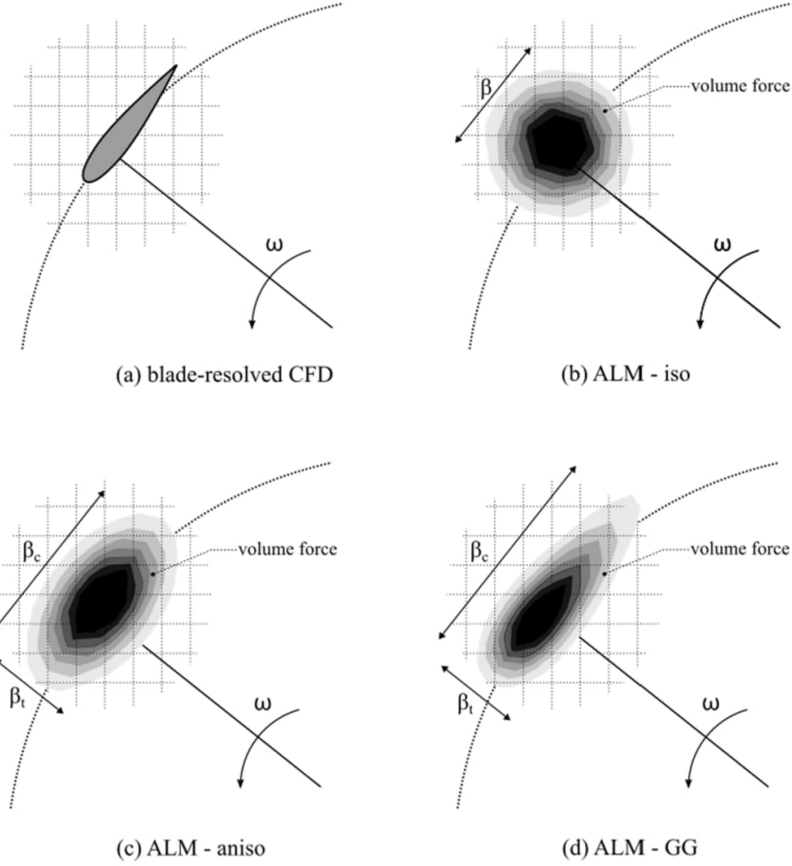


Fig. 5. (a) blade-resolved CFD illustration compared with the ALM kernel shapes employed in the present study (b) isotropic Gaussian, (c) anisotropic Gaussian, and (d) anisotropic Gaussian-Gumbel.

3.1.2. Regularization kernel

As discussed earlier, to avoid non-physical discontinuities in the solution, the forces have to be projected into the CFD domain using a smearing function. In the current work, three variations of the Gaussian function have been considered:

1- Isotropic Gaussian [13], *iso*, where the forces are distributed equally in the chord and thickness directions in a cylindrical shape. This function reads:

$$\eta(r) = \frac{1}{\beta^2 \pi} \exp \left[- \left(\frac{|r|}{\beta} \right)^2 \right] \quad (6)$$

2- Anisotropic Gaussian [32], *aniso*, where the forces are distributed in an elliptical shape using different kernel widths in chord and thickness directions; β_c and β_t , respectively. This function reads:

$$\eta(r_c, r_t) = \frac{1}{\beta_c \sqrt{\pi}} \exp \left[- \left(\frac{|r_c|}{\beta_c} \right)^2 \right] \cdot \frac{1}{\beta_t \sqrt{\pi}} \exp \left[- \left(\frac{|r_t|}{\beta_t} \right)^2 \right] \quad (7)$$

3- Anisotropic Gaussian-Gumbel [33], *GG*, which is similar to the anisotropic Gaussian. However, it has a Gumbel function applied to the chordwise direction that resembles the aerofoil physical shape. This function reads:

$$\eta(r_c, r_t) = \frac{1}{\beta_c} \exp \left[- \left(\frac{|r_c|}{\beta_c} \right) \right] \cdot \exp \left[- \exp \left[- \left(\frac{|r_c|}{\beta_c} \right) \right] \right] \cdot \frac{1}{\beta_t \sqrt{\pi}} \exp \left[- \left(\frac{|r_t|}{\beta_t} \right)^2 \right] \quad (8)$$

In these functions, r is the distance between the generic cell centroid and the actuator line, and β is the kernel width parameter. In order to nondimensionalize β , it has been related in this work to the characteristic airfoil dimension, which in this case is the chord length, c [11]. An illustration of the kernel shapes that have been used in the present work is shown in Fig. 5.

3.2. Frozen ALM

The *Frozen ALM* approach introduced by [31] has been considered in the present work as a starting point to further develop the concept as to apply it to the study of Darrieus turbines. In particular, this approach is useful as it enables the distinction between the errors due to the aerodynamic coefficients or dynamic stall model and those due to the insertion of forces in the domain. Basically, in the frozen ALM, the aerodynamic coefficients are not computed from the tabulated airfoil polar data, but are instead obtained directly from a blade-resolved CFD simulation. Hence, the uncertainties related to the quality of the polar data, dynamic stall model, and ad hoc correction models are removed.

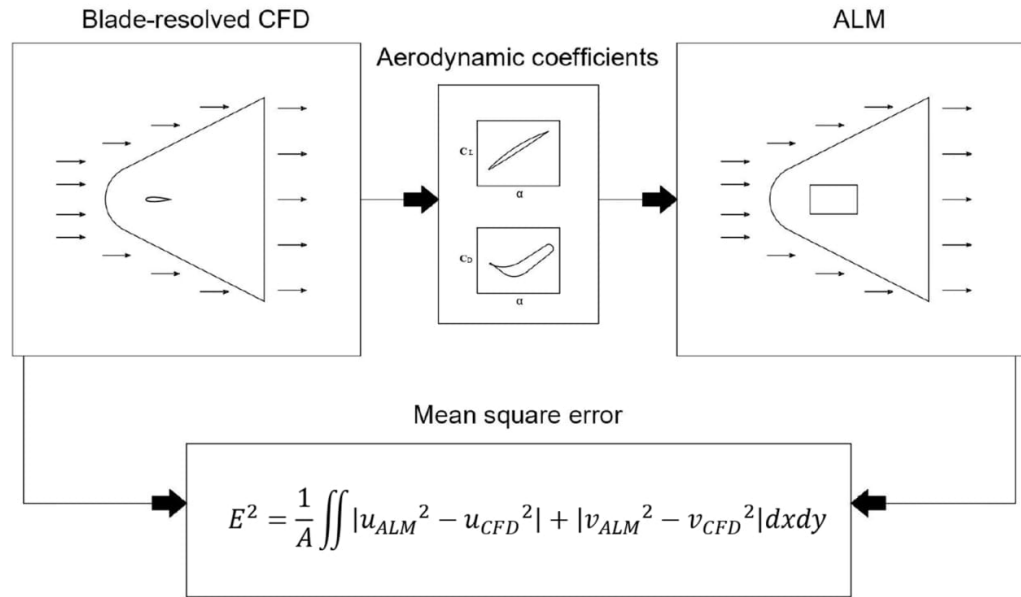


Fig. 6. Schematic representation of the kernel size calibration algorithm.

The *Frozen ALM* was first used as a calibration method for the kernel widths, β_c and β_b using a static airfoil simulation. The calibration was conducted by obtaining a mean square error, E^2 , between the flow field of the Frozen-ALM and the blade-resolved CFD, which gave an insight into the influence of the kernel width, thus allowing a wide range of interactions between different values of β_c and β_b . More in detail, E^2 was calculated between the blade-resolved CFD solution and that of the

frozen ALM, as presented in Eq. (9):

$$E^2 = \frac{1}{A} \iint_A |u_{ALM}^2 - u_{CFD}^2| + |v_{ALM}^2 - v_{CFD}^2| dx dy \quad (9)$$

where A is the domain area (except for the space allocated by the airfoil for the frozen ALM cases), u and v are the velocity components in X and Y directions, respectively. Fig. 6 shows a schematic representation

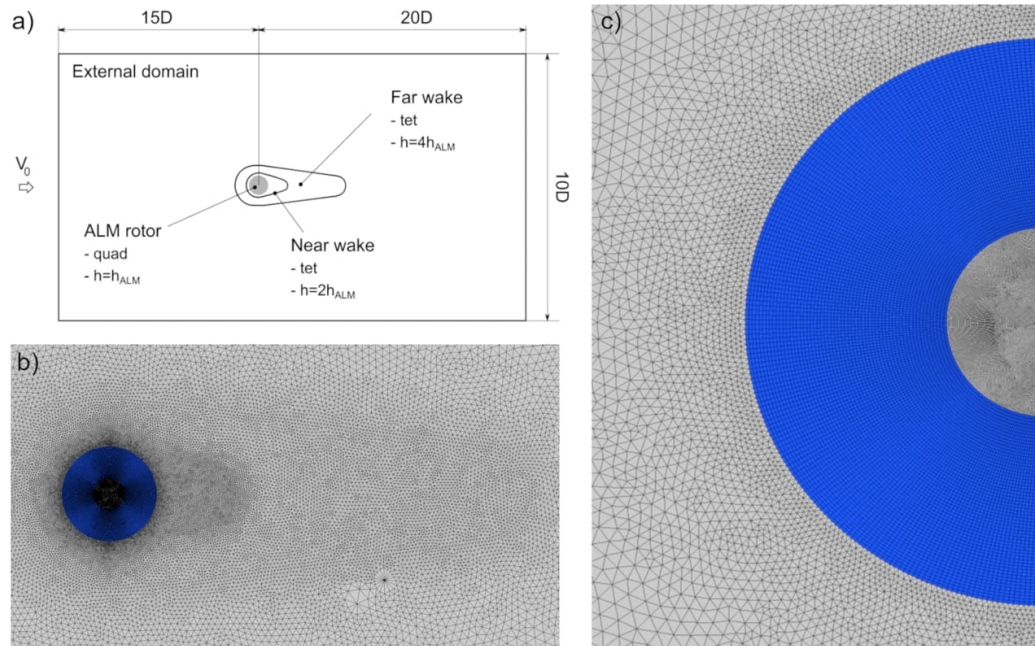


Fig. 7. a) details of the computational domain, b) overall grid, and c) ALM region grid.

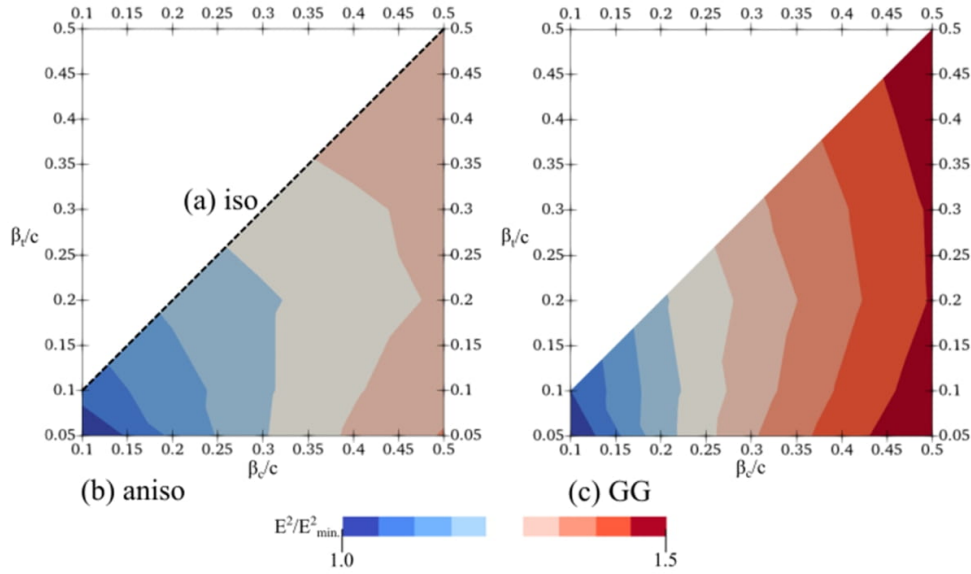


Fig. 8. Contours of the nominal mean square error E^2/E_{min}^2 of the velocity field for (a) anisotropic Gaussian and (b) anisotropic Gaussian-Gumbel obtained for different kernel widths, β_c and β_r .

of the calibration technique workflow.

After calibrating the kernel width, the *Frozen ALM* was employed in a transient case of Darrieus turbine to assess the effect of the kernel shape in the context of a transient turbine case. In this case, the loads are extracted from a blade-resolved CFD case and then sent to the ALM code to be projected into the computational grid as equivalent volume momentum sources.

3.3. CFD set-up

3.3.1. Turbine model

All two-dimensional simulations were carried out with an unsteady Reynolds-Averaged Navier-Stokes (URANS) approach via the commercial solver ANSYS® FLUENT®, in the 2020 R2 release. For turbulence closure, the *k- ω* Shear Stress Transport (SST) model was selected [51]: compared to the other models available, it provides better accuracy and stability in unsteady flow conditions such as the ones experienced by VAWT blades, with massively separated flows and strong pressure gradients [10]. For detailed information about the settings for adopted numerical schemes, convergence definition, maximum inner iterations, and residuals, please refer to [10] and [52].

3.3.2. Domain and boundary conditions

An *open-field* domain was chosen for the analysis. The corresponding dimensions, parameterized with respect to turbine diameter D (see Fig. 7), were selected according to the sensitivity analysis from [48]. This allowed minimizing the disturbance induced by the turbine on the freestream flow, justifying the application of the standard *far field* boundary conditions commonly adopted for external flows. In detail, a uniform velocity, turbulent kinetic energy k , and length scale L were applied at the inlet based on the testing conditions of the considered test case. The turbulent quantities, in particular, were estimated according to the test inlet turbulent intensity and length scale. At the outlet, uniform atmospheric pressure was imposed, while for the other patches the *symmetry* condition was adopted.

3.3.3. Grid

The computational domain was discretized with a hybrid meshing

strategy. As shown in Fig. 7, the CFD domain was divided into three separated fluid zones, according to the main features commonly found in a Darrieus turbine operating in an open-field environment:

- **Rotor ALM region:** The toroid encompassing the turbine rotor, where the forces are applied. In contrast with the rest of the domain, this region was discretized with a structured mesh, according to the requirements of the ALM. The number of elements in the azimuthal direction is directly proportional to the timestep used for the CFD simulations so that no discontinuity occurs when advancing the actuator line in the domain. The cell dimensions in the radial and spanwise direction, on the other hand, were chosen so that a cell Aspect Ratio as close as possible to 1 is achieved, at least on the turbine trajectory. This avoids the distortion of applied volume force distribution. The element size in this region is referred to in this paper as Δh and is always related to the kernel width β . A dedicated analysis of the model sensitivity to ALM region element size Δh is presented in Section 6.4;
- **Wake region:** An unstructured quad-dominant mesh was used for the drop-shaped refinement region near the rotor, which is necessary to properly capture the turbine blockage and wake development. As shown in Fig. 7, this region is divided into two separate zones: a *near-* and *far-wake* one. The former, characterized by a cell dimension h_{NW} , always taken as $2\Delta h$ comprehends the zone where the highest wake mixing is encountered. The latter, with $h_{FW} = 2h_{NW}$, serves to both resolve the far wake development and ensure a smooth transition from the rotor to the external domain;
- **External region:** The prismatic zone representing the external flow domain, discretized with tetrahedral elements.

4. Results

Initially, in Section 4.1, a calibration of the kernel width was conducted using the *Frozen ALM* approach, by comparing the flow field of a two-dimensional blade-resolved CFD for a static airfoil with that of the *Frozen ALM*. Indeed, investigating the interactions between different

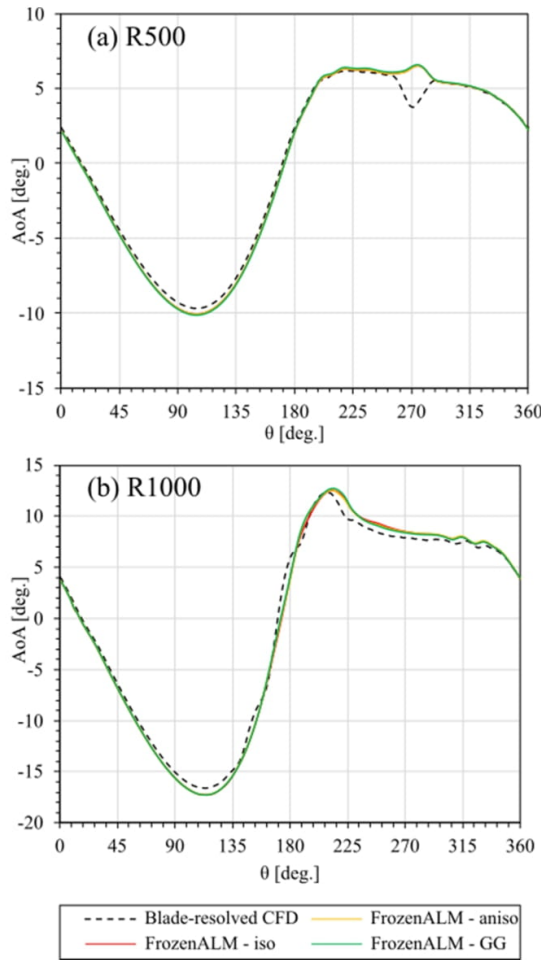


Fig. 9. Angle of attack sampling for the (a) R500 and (b) R1000 using FrozenALM of different smearing functions: iso, aniso, and GG compared to that sampled from the blade-resolved CFD.

levels of kernel sizes (from β_c or $\beta_t = 0.1c$ to β_c or $\beta_t = 0.5c$) provided an insight into its influence. Thereafter, to investigate the effect of the kernel shape and size in the light of the performance complexities of the Darrieus rotor, the FrozenALM approach was applied to a rotor case in Section 4.2. Then, in Section 4.3, using the standard ALM, different kernel widths were evaluated. In Section 4.4, the solution sensitivity to the grid size was analyzed for each of the employed kernel functions. Finally, in Section 4.5, some comparative analyses and remarks were highlighted.

An important aspect of this study is that two different rotors of different solidities were studied: R1000, which is a Darrieus Hydrokinetic Turbine (HKT), and R500, which is a Darrieus Vertical Axis Wind Turbine (VAWT). Using such an approach was useful in understanding the significance of the results in the context of different fluids and turbine geometries.

4.1. Static airfoil: Kernel calibration

To investigate the influence of the kernel width, a parametric study has been first carried out using different combinations of β_c and β_t

values. To do so, the frozen ALM approach was utilized for a static NACA0018 airfoil case at $AoA = 15^\circ$, for which the blade-resolved CFD simulation was showing the presence of flow separation. The blade forces resulting from the latter simulation were then projected into the CFD flow field using different smearing functions as discussed in Section 3.2.

Fig. 8 shows the contours of the normalized square error E^2/E_{min}^2 for different values of β_c and β_t , where E_{min}^2 is the minimum normalized square error obtained; the isotropic line is shown in Fig. 8-a. Overall, the results show that the solution is more sensitive to the kernel width in the chord direction β_c than to the thickness direction β_t . This observation agrees with the theoretical analysis conducted by Martínez-Tossas et al. [31]. Furthermore, the results show that, generally, the size of the kernel has a larger impact in the case of the GG compared to the aniso. From these results, it could be observed that using a kernel width in the thickness-wise direction β_t as half of that in the chord-wise direction β_c would be the best practice, especially while using a large value of β_c , e.g., $\beta_c > 0.3c$. The latter remark could be even more noticeable in the GG case as shown in Fig. 8-b.

4.2. Rotor simulations: Effect of kernel shape

To study the effect of the kernel shape, the FrozenALM has been employed for the R500 and the R1000 models in order to eliminate the uncertainties related to polar data. Fig. 9 shows the sampled angle of attack for FrozenALM - iso, FrozenALM - aniso, and FrozenALM - GG compared to that sampled from a blade-resolved CFD. It is important to point out here that the local velocity sampling for the blade-resolved CFD was conducted using the same LineAverage technique adopted for the ALM with the settings mentioned in Section 3.1. In the present cases of FrozenALM, kernel widths were taken as $\beta_c = 0.1c$ and $\beta_t = 0.05c$ for both the aniso and GG cases, while for the iso case, $\beta = 0.1c$ has been used. As it can be seen, all frozen ALM cases - regardless of the employed smearing function - were able to sample the AoA around the hypothetical blade almost equally, which matches as well with the AoA sampled from the blade-resolved CFD.

Fig. 10 shows the contours of the dimensionless velocity V/V_∞ for the R1000 case at different azimuth positions, θ , for the blade-resolved CFD compared to that of the FrozenALM solutions. Although the three kernel functions showed the same AoA predictions (see Fig. 9), the blade wakes resulting from the FrozenALM-iso seem to be more poorly represented compared to that of the FrozenALM-aniso and FrozenALM-GG, especially in the upstream azimuthal positions, e.g., $\theta = 60^\circ$, 120° , and 180° . The latter observation could be attributed to the narrow distribution of forces in the thickness direction for the anisotropic functions. On the other hand, the near wake of the rotor seems to be the same for all the smearing functions and matches with the blade-resolved CFD flow field.

4.3. Rotor simulations: Effect of the kernel size

To investigate the effect of the kernel width in a transient context, a sensitivity analysis has been conducted for each of the three smearing functions. For both aniso and GG, the kernel width in the thickness-wise direction was always taken as $\beta_t = 2\beta_c$ as discussed earlier in Section 4.1. Regarding the grid size, the ALM grid element size was taken as $\Delta h = \beta_t/2$ for both aniso and GG, and $\Delta h = \beta/2$ for the iso.

4.3.1. R500 rotor

Regarding the R500 VAWT rotor, Fig. 11 shows the prediction of the blade normal force coefficient C_N ($C_N = F_N/0.5\rho V_\infty^2$) and tangential force coefficient C_T ($C_T = F_T/0.5\rho V_\infty^2$) for multiple kernel widths for ALM-iso compared to that of the blade-resolved CFD and experimental results from [37]. As discussed, since a TSR of 4.5 is investigated here, the impact of dynamic stall is here negligible. Upon examination of the figure it is apparent that, compared to the blade-resolved CFD, the ALM

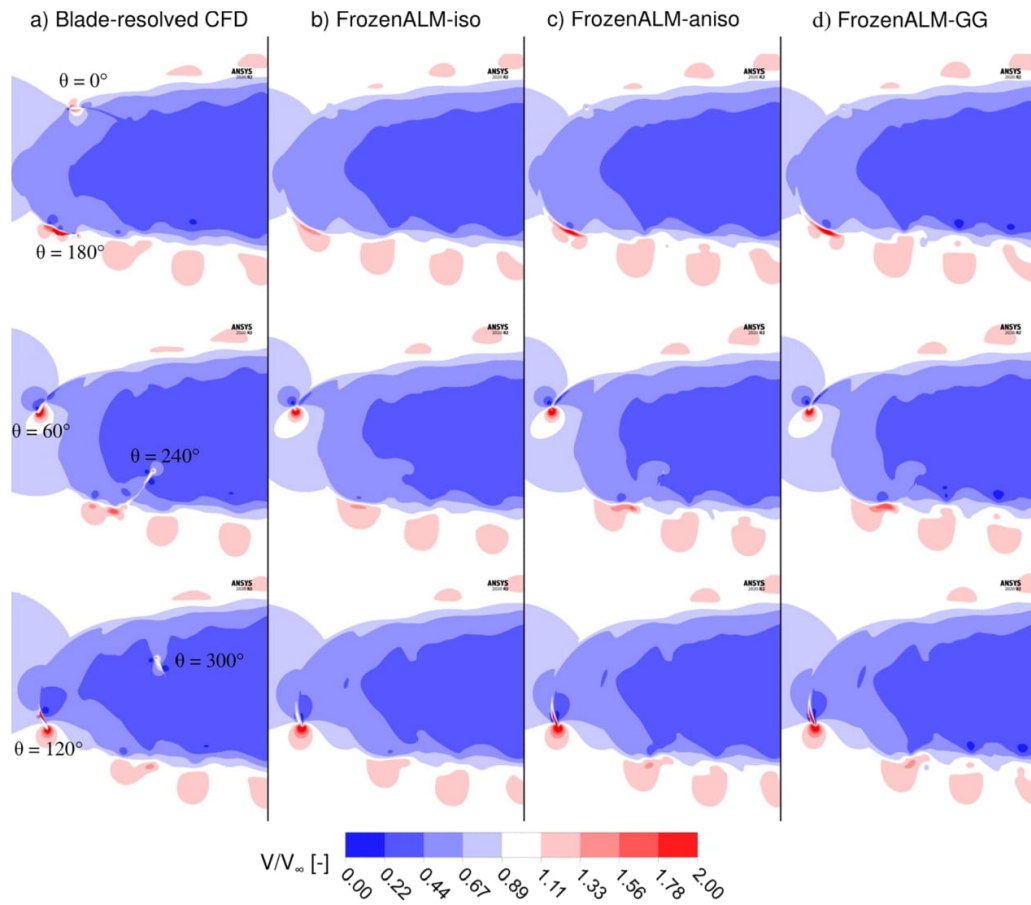


Fig. 10. Dimensionless velocity contours of R1000 rotor at various blade azimuth positions for the blade-resolved CFD compared with frozen ALM cases using *iso*, *aniso*, and *GG* smearing functions.

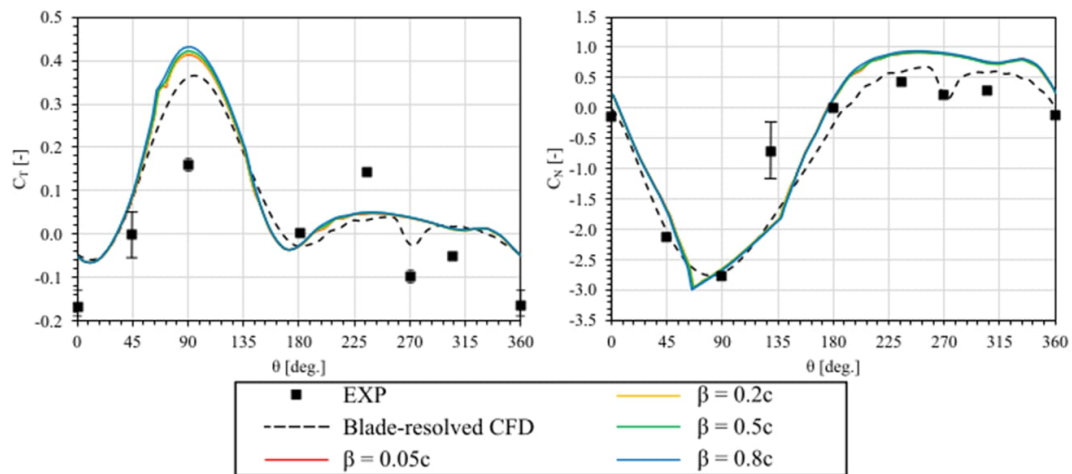


Fig. 11. Predicted variation of tangential and normal forces coefficients for a single blade of R500 over a full revolution for the blade-resolved CFD and ALM-iso using different kernel widths compared with experimental data.

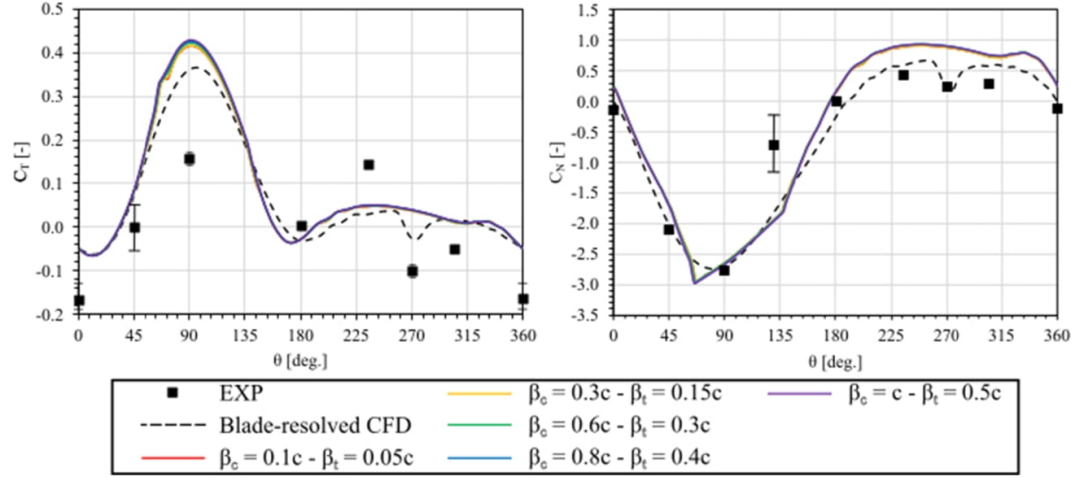


Fig. 12. Predicted variation of tangential and normal forces coefficients for a single blade of R500 over a full revolution for the blade-resolved CFD and ALM-aniso using different kernel widths compared with experimental data.

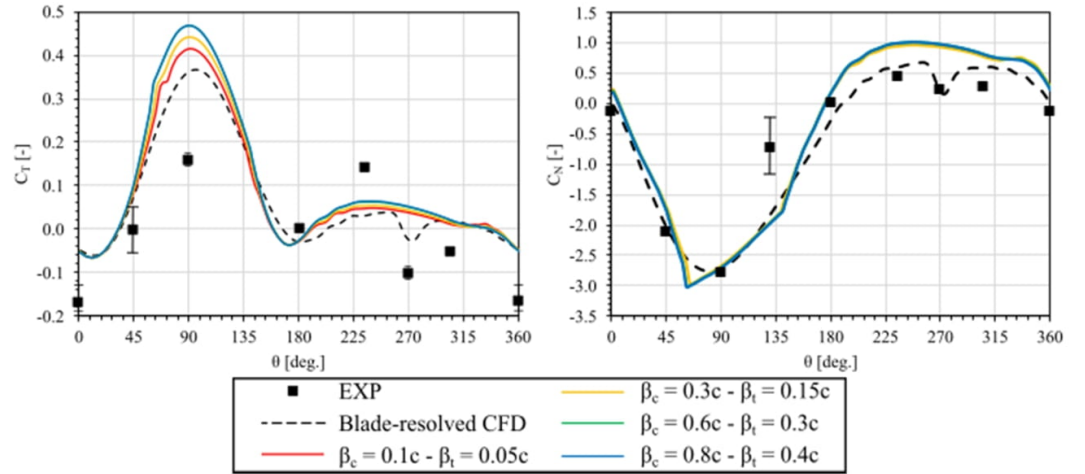


Fig. 13. Predicted variation of tangential and normal forces coefficients for a single blade of R500 over a full revolution for the blade-resolved CFD and ALM-GG using different kernel widths compared with experimental data.

was able to predict the blade's normal load during the upwind section regardless of the adopted kernel size. It must be noted that experimental data are subject to a high degree of uncertainty due to issues in the extrapolation of forces from PIV data [37]. In the downwind region, the ALM is slightly overpredicting the load. Fig. 12 shows the normal force coefficient for *ALM-aniso* using various kernel widths and compared to that of blade-resolved CFD. Considerations similar to the above can be made here, where increasing the kernel width does not significantly influence the prediction of the load. Moving to the *ALM-GG*, Fig. 13 shows that, unlike the other two smearing functions, the Gumbel function seems to be more sensitive to the width of the kernel as observed earlier in Section 4.1. Adopting larger kernel widths leads to an overprediction in the load during the upwind region at $45^\circ < \theta < 70^\circ$ and during the downwind region at $180^\circ < \theta < 315^\circ$.

4.3.2. *r1000*

Fig. 14 shows the computed *R1000* blade normal and tangential

coefficients over a full revolution for *ALM-iso* using different kernel widths compared to those of blade-resolved CFD. Overall, it can be noticed that the kernel width does not significantly affect the load predictions. More in detail, looking at the first quadrant of the blade azimuthal path ($0^\circ < \theta < 90^\circ$), all kernel widths predicted the force similarly until reaching $\theta \approx 60^\circ$, which corresponds to $AoA = 10^\circ$ (see Fig. 9), where the higher kernel widths result in a slight overestimation of the load. Thereafter, during the second azimuthal quadrant ($90^\circ < \theta < 180^\circ$), the load predictions are perfectly matched for all cases. However, during the downstream azimuth positions ($180^\circ < \theta < 360^\circ$), one can notice slight differences between the load estimation, which can be ascribed to the effect of velocity sampling within the shed vortex flow field. Generally, the vortical flow structure that is formed in the wake of the hypothetical blade during the upstream azimuthal path is convected downstream and interacts with blades during their downstream azimuthal path. The size and the intensity of those vortices are highly dependent on the adopted grid element size that is in turn a function of

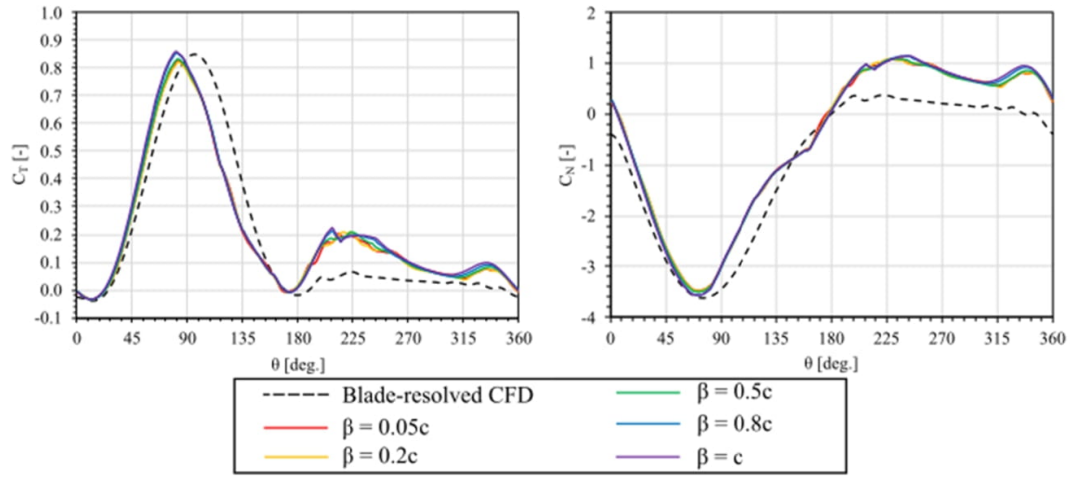


Fig. 14. Predicted tangential force variation for a single blade of R1000 over a full revolution for the blade-resolved CFD and ALM-iso using different kernel widths compared with experimental data.

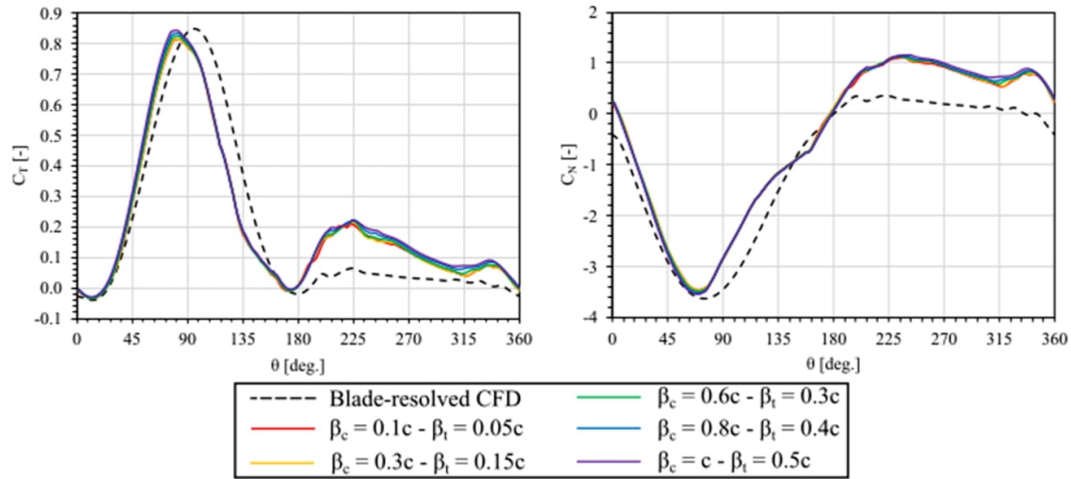


Fig. 15. Predicted tangential force variation for a single blade over a full revolution for the blade-resolved CFD compared with ALM-aniso using different kernel widths.

the kernel size. A detailed analysis is conducted for the flow field of ALM with different grid sizes and kernel shapes in Section 4.5.

Fig. 15 shows the computed normal and tangential coefficients for the R1000 using ALM-aniso with different kernel widths. From a perusal of the figure, one can note that the results are quite similar to those obtained from the ALM-iso, showing no significant effect of changing the kernel width. Moving to the ALM-GG, Fig. 16 reports the estimated normal and tangential force coefficients using different kernel sizes for the R1000. It can be observed that this smearing function is much more sensitive to the kernel size compared to the other circular and parabolic functions. The results of the ALM-GG show that differences in the load estimation start during the first quadrant and reach their maximum at $\theta \approx 75^\circ$, where the load peaks. However, within the azimuthal range $100^\circ < \theta < 190^\circ$, the load predictions are almost the same for all the kernel widths. After that, and during almost the whole downstream azimuthal path ($190^\circ < \theta < 360^\circ$), the loads are significantly different, especially at $\beta_c > 0.2c$, where the trend is that the larger the kernel size, the larger the

force estimation. This was expected considering the sensitivity of the additional Gumbel function to the width of the kernel as discussed in Section 4.1.

4.4. Effect of grid size

To investigate the influence of ALM grid size Δh for each of the kernel shapes, a dedicated grid sensitivity analysis has been conducted for the R1000. Table 3 reports the details of the investigated set-ups. It is common practice in ALM to relate Δh with the size of the kernel β [13]. Based on previous literature on HAWTs, it is known that a ratio of $\Delta h = \beta/2$ would guarantee a stable and robust solution. In the present analysis, the question was which kernel width should be related to the grid size while using non-isotropic functions, e.g., β_c or β_t . Thus, several models were tested using the same kernel width while varying the size of the grid element size. For ALM-iso, $\beta = 0.5c$ has been used, and for ALM-aniso and ALM-GG, $\beta_c = 0.6c - \beta_t = 0.3c$ and $\beta_c = 0.2c - \beta_t = 0.1c$,

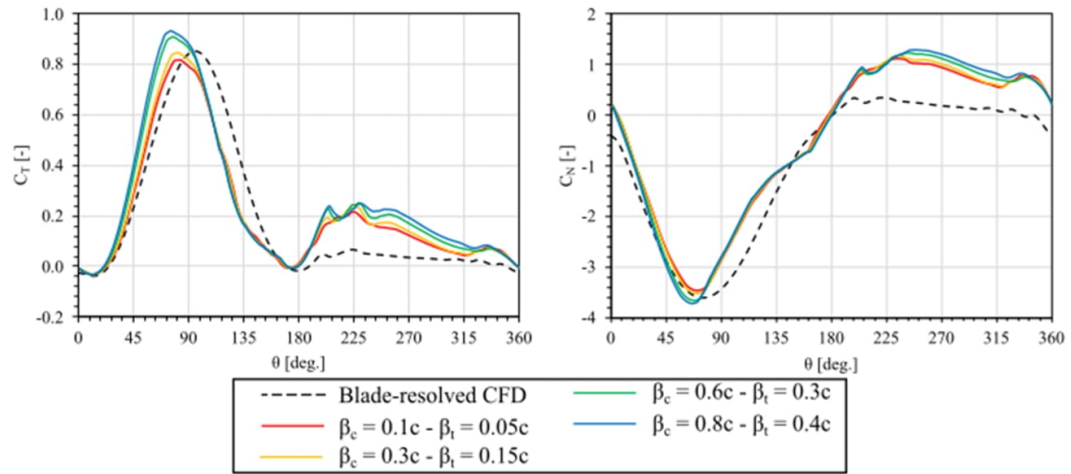


Fig. 16. Predicted tangential force variation for a single blade over a full revolution for the blade-resolved CFD compared with ALM-GG using different kernel widths.

Table 3
Grid senestivity analysis set-ups.

Set-up		Δh [mm]	Number of elements
iso - $\beta = 0.5c$	$\Delta h = \beta/2$	62.5	11,860
	$\Delta h = \beta/1.5$	83.3	10,763
	$\Delta h = \beta$	125	9854
aniso - $\beta_c = 0.6c, \beta_t = 0.3c$	$\Delta h = \beta_t/2$	37.5	16,764
	$\Delta h = \beta_t/1.5$	50	13,392
	$\Delta h = \beta_t$	75	11,017
	$\Delta h = 1.5\beta_t$	112.5	9991
	$\Delta h = 2\beta_t$	150	9559
GG - $\beta_c = 0.2c, \beta_t = 0.1c$	$\Delta h = \beta_t/2$	12.5	72,819
	$\Delta h = \beta_t/1.5$	16.7	44,655
	$\Delta h = \beta_t$	25	25,086

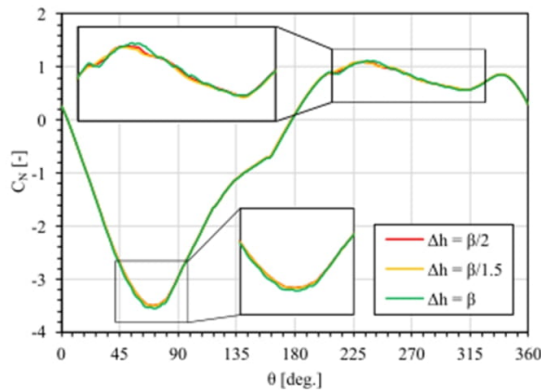


Fig. 17. Variation of the predicted normal force coefficient of $R1000$ for the ALM-iso ($\beta = 0.5c$) using different grid sizes.

respectively.

Fig. 17 shows the results of the normal force for the three evaluated kernel shapes using different grid sizes. It can be seen that for the ALM-iso, using $\Delta h = \beta/2$ and $\Delta h = \beta/1.5$ produces the same normal force profiles, while when increasing Δh so as to be the same size of the kernel, one can notice the instability of the predicted force profile at the

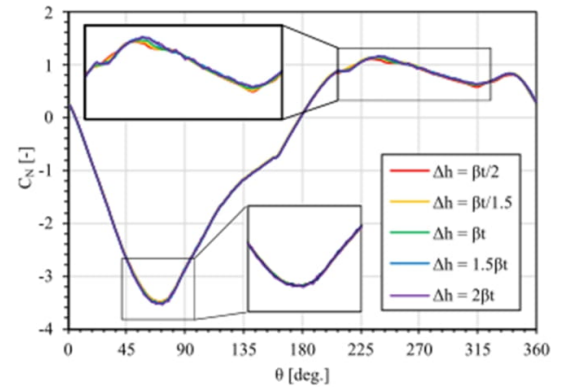


Fig. 18. Variation of the predicted normal force coefficient of $R1000$ for the ALM-aniso ($\beta_c = 0.6c - \beta_t = 0.3c$) using different grid sizes.

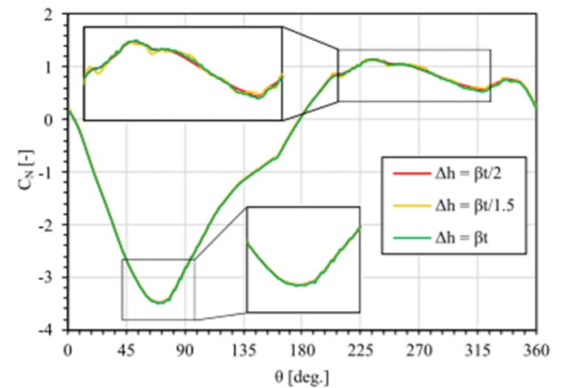


Fig. 19. Variation of the predicted normal force coefficient of $R1000$ for the ALM-GG ($\beta_c = 0.2c - \beta_t = 0.1c$) using different grid sizes.

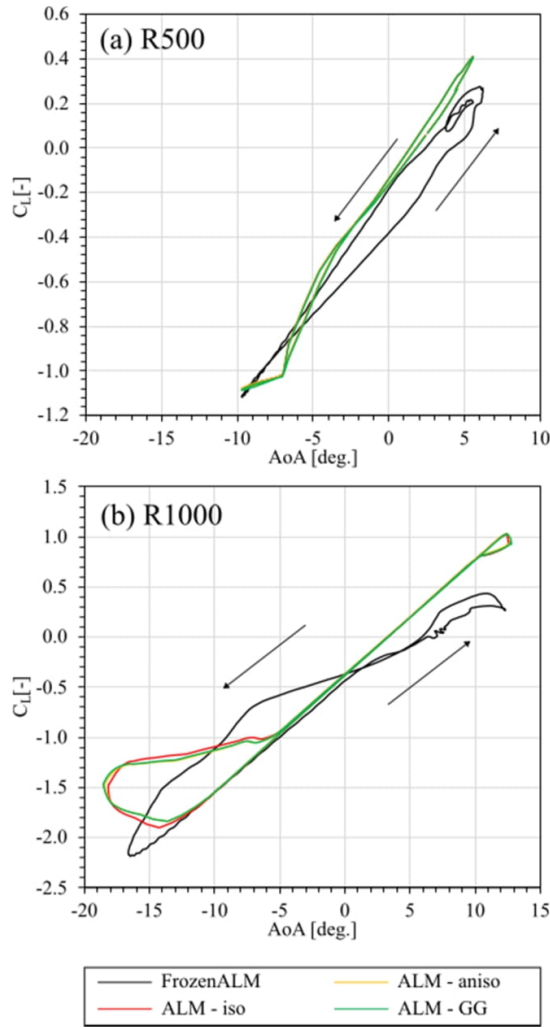


Fig. 20. Hysteresis loop of lift coefficients obtained from *FrozenALM* compared to that of the ALM variations using different kernel shapes.

positions where flow separation is expected, e.g., $60^\circ < \theta < 90^\circ$ and $240^\circ < \theta < 300^\circ$.

For *ALM-aniso*, the ALM element size Δh has been related first to the kernel in the thickness direction β_b , and as can be seen from Fig. 18, the performance prediction results seem to be unaffected by the grid size until Δh reaches $1.5\beta_b$, where the instabilities in the normal force profile start to appear and it increases significantly at $\Delta h = 2\beta_b$ which corresponds in this case to $\Delta h = \beta_c$. On the other hand, for *ALM-GG*, the load predictions seem to be more sensitive to the mesh size, as seeable from Fig. 19, at $\Delta h = \beta_b$, instabilities in the load predictions could be seen clearly at $75^\circ < \theta < 100^\circ$ and $200^\circ < \theta < 330^\circ$.

4.5. Comparative analyses and additional remarks

Fig. 20 shows the resulting hysteresis loops of lift coefficient, C_L , for the three variations of the ALM using kernel functions of *iso*, *aniso*, and *GG*, compared to those obtained from the *FrozenALM*, which are basically the same as those of the blade-resolved CFD. As can be seen in

Fig. 20-a for *R500*, one can observe that the kernel shape does not affect the prediction of C_L , and all the ALM variations show the exact same C_L , which almost matches that of the *FrozenALM*. However, the ALM was not able to reproduce the unsteady effects which can be noticed around $\text{AoA} \approx 5^\circ$ for the lift coefficient of the *FrozenALM*. Moving to *R1000* in Fig. 20-b, significant deviations could be noticed between the C_L of the *FrozenALM* and that obtained from the standard ALM variations. Also, it shows that the kernel shape does not influence the C_L predictions, despite the slight differences that could be noticed at high angles of attack, e.g., $\text{AoA} < -10^\circ$, which are attributed to the implementation of the dynamic stall model inside the code. Generally, comparing the C_L results of the *FrozenALM* with that of the standard ALM shows that the presence of the dynamic stall phenomenon plays the most important role in producing a reliable and accurate load prediction.

Fig. 21 shows the contours of dimensionless vorticity of *R1000* for different ALM cases with different kernel shapes, widths, and grid sizes compared to that of the blade-resolved CFD. From Fig. 21-b and 21-d, it can be noticed that the flow field of the *ALM-aniso* is extremely sensitive to the width of the kernel, despite that both cases resulted in the same load predictions (see Fig. 15). Moreover, comparing the latter cases to the blade-resolved CFD presented in Fig. 21-a, and focusing on the shear layers that define the wake borders, one can observe that the *ALM-aniso* with the smallest kernel width ($\beta_c = 0.1c - \beta_t = 0.05c$) reproduced the diffusion of the red anticlockwise vortices more accurate compared to that with the larger kernel width ($\beta_c = 0.6c - \beta_t = 0.3c$). However, the blue clockwise vortices tend to be more compact in the case of the smaller kernel width compared to that of the blade-resolved CFD. Conversely, the larger kernel width reproduced the convection of these vortices more similar to that of the blade-resolved CFD.

Moving to the *ALM-iso*, Fig. 21-e and 21-f bring to the same conclusion that has been observed for the *ALM-aniso*: despite using $\beta = 0.05c$ and $\beta = 0.5c$ showed the same load predictions (see Fig. 14), the flow fields of both cases are completely different, showing more compact vortices in the wake region in the case of smaller kernel width. The latter observation is attributed more to the grid size than to either the kernel shape or size. To elaborate on that, one can compare the cases in Fig. 21-c and 21-d or in Fig. 21-b and 21-f. The former shows the *ALM-aniso* of the same kernel widths ($\beta_c = 0.6c - \beta_t = 0.3c$) but with different grid element sizes in the ALM region (Δh), while the latter shows the ALM results of the same grid size but employing two different kernel shapes, *iso* and *aniso*. Interestingly, the ALM is able to reproduce the wake flow field similarly for both *ALM-iso* and *ALM-aniso* if the same grid element size is used. Whereas, in the case of employing the same kernel shape, but with a different grid element size, the flow field in the wake is significantly different.

From the latter analysis, one can say that the effect of the shape of the kernel is not significant in either the load prediction or the flow field; conversely, it can be said that the grid size plays the most important role in reproducing the flow field wake.

5. Discussion

While the amount of results showed in Section 4 can make the reader lose the narrative path, the information resulting from a general interpretation of them is quite clear and of capital relevance in the field of Darrieus VAWT simulation.

Overall, the comparison with both experiments and blade-resolved CFD showed that the current implementation of the ALM [23] is adequate to predict both rotor performance and flow field with an acceptable level of accuracy at medium and high tip-speed ratios, provided that the correct aerodynamic data is fed into the code, and the grid is fine enough to capture the complexities of the wake flow structures.

The shape of the kernel, which has been long deemed to represent a major source of discrepancies [23] seems not to be relevant in the case of Darrieus vertical axis turbines. This differs from the ALM for HAWTs, where previous works have demonstrated the role of non-isotropic

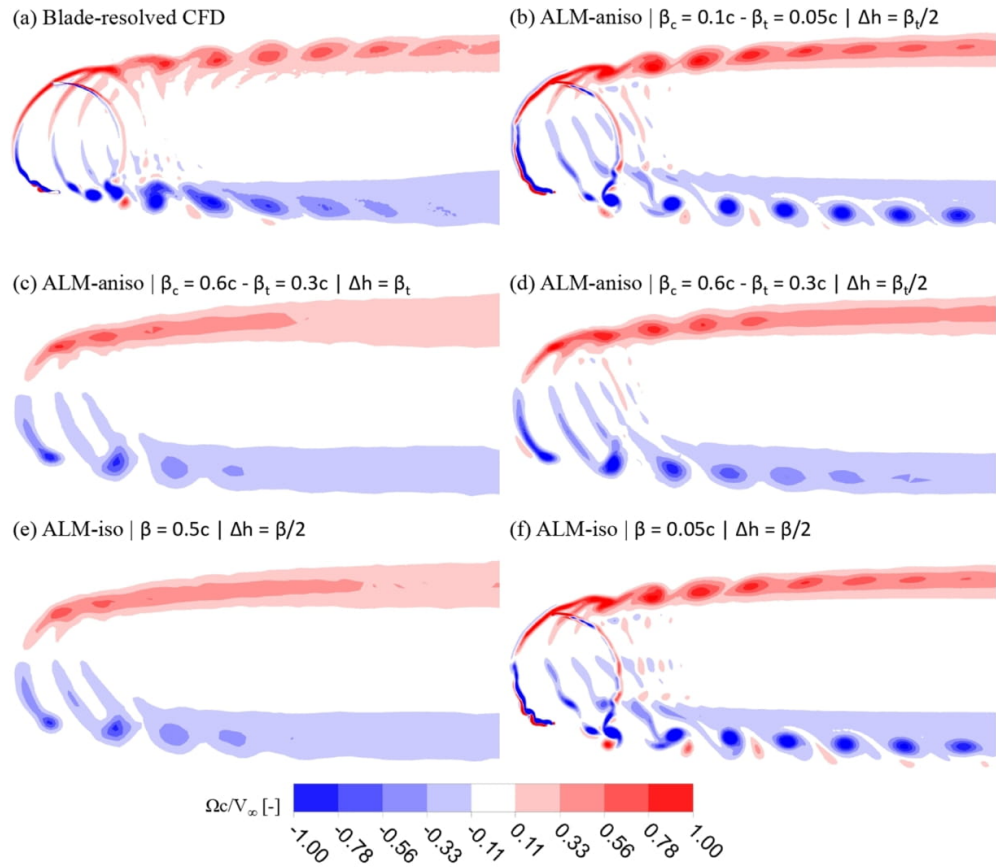


Fig. 21. Dimensionless vorticity contours of *R1000* for various ALM cases using different kernel shapes, sizes, and grid sizes compared to that of the blade-resolved CFD.

kernel shapes in reproducing the wake more accurately [28 29 30]. In VAWTs the effect of the kernel size on the other hand, which has been deemed a source of discrepancies in reproducing the shed vortical structures of the Darrieus turbine wake [18], seems to be mutually dependent on the grid size but does not itself affect the accuracy of the solution, except in the case of employing *GG* kernel function. As remarked in Fig. 21, using *aniso* kernel shape with a relatively large kernel size, e.g., $\beta_c = 0.6c$ and $\beta_t = 0.3c$, and with a proper grid refinement, can reproduce the wake flow field with good accuracy compared to blade-resolved CFD. Indeed, it is important to note that the sampling strategy for the blade local flow field plays a crucial role in minimizing the influence of the kernel shape or size on the ALM solution. For example, the conventional ALM sampling method which has been implemented either in the original work of Sørensen and Shen [11] or in most of the works that follow, is based on sampling the velocity directly along the actuator line itself, in another word, at the center of the bound vortex, where the blade local flow effects are not seen. The latter method could be sensitive to the kernel size as noted by Martínez et al. [53]. However, the *LineAverage* strategy, employed in this work, does not have this limitation, since the local velocity is sampled as an integral average along a closed line around the actuator line.

Moving toward the influence of the aerodynamic coefficients data, it has been demonstrated that the ALM is capable of generating an extremely accurate wake flow field, close to that of the blade-resolved CFD (see Fig. 10) if the code is fed with the correct polar data. The

FrozenALM solution showed that the aerodynamic coefficients play the most important role in the ALM solution accuracy. In particular, in the case of the *R1000* at $TSR = 2.5$, where dynamic stall plays an important role, the lift coefficient results showed there are significant deviations between the lift estimated by the ALM code and that used for the *FrozenALM* case. Accordingly, a better dynamic stall model is indeed the real key factor that could lead to an improvement of the ALM accuracy.

6. Conclusions

The present work showed a dedicated assessment of the role of the *input polar data*, *dynamic stall model*, and the *force smearing function* in ALM in the context of the Darrieus rotor.

To separate the effects, the *FrozenALM* approach was developed, and implemented for the first time on Darrieus turbines. This is based on feeding the ALM with aerodynamic forces extracted directly from blade-resolved CFD, which allowed evaluating the impact of the kernel shape, while blocking the influence of polar data quality and dynamic stall modelling. To support the analyses, two rotors were analyzed, i.e., the *R500*, a VAWT with a low solidity of 0.12 operating at stable conditions at TSR of 4.5, and the *R1000*, a HKT with a medium solidity of 0.25 operating at its optimum TSR of 2.5. The results of the *FrozenALM* showed that the kernel shape does not significantly influence the capabilities of the ALM to either predict the blade loads or to reproduce the wake flow field. Rather, it showed that, the ALM is able to generate

extremely accurate solutions of the flow field, almost equivalent to blade-resolved CFD, if the aerodynamic forces are provided correctly.

Using the standard ALM, three different kernel functions have been tested in different sizes: (a) *iso* which distributes the forces in a circular shape, (b) *aniso* which has a parabolic shape, and (c) *GG* which also takes a parabolic shape but with a Gumbel function in the chordwise direction, making the distribution closer to that of the airfoil. It has been concluded that the size of the kernel does not significantly affect the solution, except in the case of using *GG* function, where the additional Gumbel function is very sensitive to the width of the kernel. Thereafter, an assessment of the grid size influence has been conducted for the three kernel shapes, showing that, regardless of the kernel shape and size, the grid resolution is the key factor that significantly influences the quality of reproducing the flow field. Further, the grid sensitivity analysis showed that using ALM grid size, $\Delta h = \beta/2$ is a best practice while using *iso* kernel, and $\Delta h = \beta/2$ while using *aniso* or *GG* kernels.

Finally, the analysis of the two rotors under two different operating conditions allowed evaluating the role of the dynamic stall model in the accuracy of the ALM. The ALM load estimation for the *R500* (no dynamic stall) was much closer to the blade-resolved CFD compared to that of the *R1000* (dynamic stall massively present). Upon comparing the aerodynamic coefficients from the *ProzenALM* and the standard ALM, it has been concluded that the cases of large dynamic stall presence resulted in a significant mismatch in the lift coefficient data compared to those derived from blade-resolved CFD. Thus, a better dynamic stall model is indeed the real key factor that could lead to an improvement in the ALM accuracy. To this end, further research will be devoted at developing a dynamic stall model that could account for the special characteristics of Darrieus rotors. This would make the ALM more reliable, especially in dealing with advanced applications such as floating offshore, multiple rotors, and urban applications.

CRedit authorship contribution statement

Omar S. Mohamed: Methodology, Software, Validation, Formal analysis, Investigation, Data curation, Writing – original draft, Visualization. **Pier Francesco Melani:** Methodology, Software, Writing – review & editing, Visualization. **Francesco Balduzzi:** Conceptualization, Methodology, Writing – review & editing. **Giovanni Ferrara:** Resources, Supervision, Project administration, Funding acquisition. **Alessandro Bianchini:** Conceptualization, Methodology, Investigation, Resources, Data curation, Writing – review & editing, Supervision, Project administration, Funding acquisition.

Declaration of Competing Interest

The authors declare that they have no known competing financial interests or personal relationships that could have appeared to influence the work reported in this paper.

Data availability

Data will be made available on request.

References

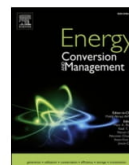
- Melani PF, Balduzzi F, Ferrara G, Bianchini A. How to extract the angle attack on airfoils in cycloidal motion from a flow field solved with computational fluid dynamics? Development and verification of a robust computational procedure. *Energy Convers Manag* Nov. 2020;223:113284. <https://doi.org/10.1016/j.enconman.2020.113284>.
- Mohamed OS, Elbaz AMR, Bianchini A. A better insight on physics involved in the self-starting of a straight-blade Darrieus wind turbine by means of two-dimensional Computational Fluid Dynamics. *J Wind Eng Ind Aerodyn* Nov. 2021;218:104793. <https://doi.org/10.1016/j.jweia.2021.104793>.
- O. S. Mohamed, A. Ibrahim, and A. M. R. El Baz, 'CFD Investigation of the Multiple Rotors Darrieus Type Turbine Performance', presented at the ASME Turbo Expo 2019: Turbomachinery Technical Conference and Exposition, Nov. 2019. 10.1115/ GT2019-91491.
- Paulsen US, Madsen HA, Hattel JH, Baran I, Nielsen PH. Design Optimization of a 5 MW Floating Offshore Vertical-axis Wind Turbine. *Energy Procedia* Jan. 2013;35: 22–32. <https://doi.org/10.1016/j.egypro.2013.07.155>.
- Balduzzi F, Bianchini A, Carnevale EA, Ferrari L, Magnani S. Feasibility analysis of a Darrieus vertical-axis wind turbine installation in the rooftop of a building. *Appl Energy* Sep. 2012;97:921–9. <https://doi.org/10.1016/j.apenergy.2011.12.008>.
- Balduzzi F, Melani PF, Soraperra G, Brighenti A, Battisti L, Bianchini A. Some design guidelines to adapt a Darrieus vertical axis turbine for use in hydrokinetic applications. *E3S Web Conf* 2021;312:08017. <https://doi.org/10.1051/e3sconf/202131208017>.
- Ferreira CJS, Bijl H, van Bussel G, van Kuik G. Simulating Dynamic Stall in a 2D VAWT: Modeling strategy, verification and validation with Particle Image Velocimetry data. *J Phys Conf Ser* Jul. 2007;75:012023. <https://doi.org/10.1088/1742-6596/75/1/012023>.
- M. Raciti Castelli, A. Englaro, and E. Benini, 'The Darrieus wind turbine: Proposal for a new performance prediction model based on CFD', *Energy*, vol. 36, no. 8, pp. 4919–4934, Aug. 2011, 10.1016/j.energy.2011.05.036.
- A. Bianchini, F. Balduzzi, G. Ferrara, G. Persico, V. Dossena, and L. Ferrari, 'A Critical Analysis on Low-Order Simulation Models for Darrieus Vavts: How Much Do They Pertain to the Real Flow?', *J. Eng. Gas Turbines Power*, vol. 141, no. 1, Sep. 2018, 10.1115/1.4040851.
- Balduzzi F, Bianchini A, Maleci R, Ferrara G, Ferrari L. Critical issues in the CFD simulation of Darrieus wind turbines. *Renew Energy* 2016;85:419–35. <https://doi.org/10.1016/j.renene.2015.06.048>.
- J. N. Sorensen and W. Z. Shen, 'Numerical Modeling of Wind Turbine Wakes', *J. Fluids Eng.*, vol. 124, no. 2, pp. 393–399, May 2002, 10.1115/1.1471361.
- Shen WZ, Sorensen JN, Mikkelsen R. Tip Loss Correction for Actuator/Navier-Stokes Computations. *J Sol Energy Eng Apr.* 2005;127(2):209–13. <https://doi.org/10.1115/1.1850488>.
- Shives M, Crawford C. Mesh and load distribution requirements for actuator line CFD simulations. *Wind Energy* 2013;16(8):1183–96. <https://doi.org/10.1002/we.1546>.
- Troldborg N, Sorensen JN, Mikkelsen R. Numerical simulations of wake characteristics of a wind turbine in uniform inflow. *Wind Energy* 2010;13(1): 86–99. <https://doi.org/10.1002/we.345>.
- 'Large-eddy simulation of a very large wind farm in a stable atmospheric boundary layer: Physics of Fluids: Vol 23, No 6'. <https://aip.scitation.org/doi/10.1063/1.3589857> (accessed May 23, 2022).
- S. Shamsoddin and F. Porté-Agel, 'Large Eddy Simulation of Vertical Axis Wind Turbine Wakes', *Energies*, vol. 7, no. 2, Art. no. 2, Feb. 2014, 10.3390/en7020890.
- P. Bachant, A. Goude, and M. Wosnik, 'Actuator line modeling of vertical-axis turbines', arXiv:1605.01449, Sep. 2018. 10.48550/arXiv.1605.01449.
- Mendoza V, Bachant P, Ferreira C, Goude A. Near-wake flow simulation of a vertical axis turbine using an actuator line model. *Wind Energy* 2019;22(2): 171–88. <https://doi.org/10.1002/we.2277>.
- R. Zhao, A. C. W. Creech, A. G. L. Borthwick, V. Venugopal, and T. Nishino, 'Aerodynamic Analysis of a Two-Bladed Vertical-Axis Wind Turbine Using a Coupled Unsteady RANS and Actuator Line Model', *Energies*, vol. 13, no. 4, Art. no. 4, Jan. 2020, 10.3390/en13040776.
- V. Mendoza and A. Goude, 'Validation of Actuator Line and Vortex Models Using Normal Forces Measurements of a Straight-Bladed Vertical Axis Wind Turbine', *Energies*, vol. 13, no. 3, Art. no. 3, Jan. 2020, 10.3390/en13030511.
- R. E. Sheldahl and P. C. Klimas, 'Aerodynamic characteristics of seven symmetrical airfoil sections through 180-degree angle of attack for use in aerodynamic analysis of vertical axis wind turbines', Sandia National Labs., Albuquerque, NM (USA), SAND-80-2114, Mar. 1981. 10.2172/6548367.
- Drela M. XFOIL: An Analysis and Design System for Low Reynolds Number Airfoils. University of Notre Dame Jun. 1989. https://doi.org/10.1007/978-3-642-84010-4_1.
- Melani PF, Balduzzi F, Ferrara G, Bianchini A. Tailoring the actuator line theory to the simulation of Vertical-Axis Wind Turbines. *Energy Convers Manag* Sep. 2021; 243:114422. <https://doi.org/10.1016/j.enconman.2021.114422>.
- Melani PF, Balduzzi F, Bianchini A. Simulating tip effects in vertical-axis wind turbines with the actuator line method. *J Phys Conf Ser* May 2022;2265(3): 032028. <https://doi.org/10.1088/1742-6596/2265/3/032028>.
- A. Bianchini et al., 'An Experimental and Numerical Assessment of Airfoil Polars for Use in Darrieus Wind Turbines: Part 1 – Flow Curvature Effects', presented at the ASME Turbo Expo 2015: Turbine Technical Conference and Exposition, Aug. 2015. 10.1115/ GT2015-42284.
- Bianchini A, Balduzzi F, Ferrara G, Ferrari L. Aerodynamics of Darrieus Wind Turbines Airfoils: The Impact of Pitching Moment. *J Eng Gas Turbines Power* 2016; 139(4):Nov. <https://doi.org/10.1115/1.4034940>.
- Bianchini A, Balduzzi F, Ferrara G, Ferrari L. Critical Analysis of Dynamic Stall Models in Low-Order Simulation Models For Vertical-Axis Wind Turbines. *Energy Procedia* Nov. 2016;101:488–95. <https://doi.org/10.1016/j.egypro.2016.11.062>.
- Martínez-Tossas LA, Churchfield MJ, Meneveau C. A Highly Resolved Large-Eddy Simulation of a Wind Turbine using an Actuator Line Model with Optimal Body Force Projection. *J Phys Conf Ser* Sep. 2016;753:082014. <https://doi.org/10.1088/1742-6596/753/8/082014>.
- Cormier M, Wehling P, Lutz T. Evaluation of the Effects of Actuator Line Force Smearing on Wind Turbines Near-Wake Development. *J Phys Conf Ser* May 2021; 1934(1):012013. <https://doi.org/10.1088/1742-6596/1934/1/012013>.
- 'An Advanced Actuator Line Method for Wind Energy Applications and Beyond', *AIAA SciTech Forum*. <https://arc.aiaa.org/doi/10.2514/6.2017-1998> (accessed Jun. 13, 2022).

- [31] 'Optimal smoothing length scale for actuator line models of wind turbine blades based on Gaussian body force distribution - Martínez-Tossas - 2017 - Wind Energy - Wiley Online Library'. <https://onlinelibrary.wiley.com/doi/10.1002/we.2081> (accessed May 23, 2022).
- [32] 'An Advanced Actuator Line Method for Wind Energy Applications and Beyond | AIAA SciTech Forum'. <https://arc.aiaa.org/doi/10.2514/6.2017-1998> (accessed May 23, 2022).
- [33] Schollenberger M, Lutz T, Krämer E. 'Boundary Condition Based Actuator Line Model to Simulate the Aerodynamic Interactions at Wingtip Mounted Propellers', in *New Results in Numerical and Experimental Fluid Mechanics XII*. Cham 2020: 608–18. https://doi.org/10.1007/978-3-030-25253-3_58.
- [34] Melani PF, Balduzzi F, Bianchini A. A Robust Procedure to Implement Dynamic Stall Models Into Actuator Line Methods for the Simulation of Vertical-Axis Wind Turbines. *J Eng Gas Turbines Power* 2021;143(11):Sep. <https://doi.org/10.1115/1.4051909>.
- [35] Melani PF, Balduzzi F, Ferrara G, Bianchini A. Development of a desmodromic variable pitch system for hydrokinetic turbines. *Energy Convers Manag* Dec. 2021; 250:114890. <https://doi.org/10.1016/j.enconman.2021.114890>.
- [36] G. Tescione, D. Ragni, C. He, C. J. Simão Ferreira, and G. J. W. van Bussel, 'Near wake flow analysis of a vertical axis wind turbine by stereoscopic particle image velocimetry', *Renew. Energy*, vol. 70, pp. 47–61, Oct. 2014, 10.1016/j.renene.2014.02.042.
- [37] Casteleim D, Tescione G, Ragni D, Simao Ferreira C, Gaunaa M. In: Creating a benchmark of Vertical Axis Wind Turbines in Dynamic Stall for validating numerical models. 0 vols., American Institute of Aeronautics and Astronautics; 2015. <https://doi.org/10.2514/6.2015-0723>.
- [38] P. F. Melani, F. Balduzzi, L. Brandetti, C. J. Simão Ferreira, and A. Bianchini, 'An experimental and numerical analysis of the dynamic variation of the angle of attack in a vertical-axis wind turbine', presented at the TORQUE 2020, Delft, Netherlands, 2020.
- [39] Ferreira CS, Madsen HA, Barone M, Roscher B, Deglaire P, Arduin I. Comparison of aerodynamic models for Vertical Axis Wind Turbines. *J Phys Conf Ser Jun*. 2014; 524:012125. <https://doi.org/10.1088/1742-6596/524/1/012125>.
- [40] Jost E, Klein L, Leipprand H, Lutz T, Krämer E. Extracting the angle of attack on rotor blades from CFD simulations. *Wind Energy* 2018;21(10):807–22. <https://doi.org/10.1002/we.2196>.
- [41] C. de Boor, *A Practical Guide to Splines*. New York: Springer-Verlag, 1978. Accessed: Nov. 23, 2020. [Online]. Available: <https://www.springer.com/gp/book/9780387953663>.
- [42] 'Correlation-Based Transition Modeling for Unstructured Parallelized Computational Fluid Dynamics Codes | AIAA Journal'. <https://arc.aiaa.org/doi/10.2514/1.42362> (accessed May 23, 2022).
- [43] Viterna LA, Janetzke DC. Theoretical And Experimental Power From Large Horizontal-Axis Wind Turbines. *NASA Tech Memo* 1982.
- [44] Marten D, et al. Effects of airfoil's polar data in the stall region on the estimation of Darrieus wind turbines performance. *J Eng Gas Turbines Power* 2016;139(2): 022606–9. <https://doi.org/10.1115/1.4034326>.
- [45] Massé B. Description de deux programmes d'ordinateur pour le calcul des performances et des charges aérodynamiques pour les éoliennes à axe vertical. technical report IREQ-2379 1981.
- [46] R. E. Gormont, 'A Mathematical Model of Unsteady Aerodynamics and Radial Flow for Application to Helicopter Rotors', BOEING VERTOL CO PHILADELPHIA PA, May 1973. Accessed: Nov. 16, 2020. [Online]. Available: <https://apps.dtic.mil/sti/citations/AD0767240>.
- [47] Migliore PG, Wolfe WP, Fanucci JB. Flow Curvature Effects On Darrieus Turbine Blade Aerodynamics. *J Energy* 1980;4(2):49–55. <https://doi.org/10.2514/3.62459>.
- [48] P. F. Melani, P. Schito, and G. Persico, 'Experimental assessment of an ACL simulation tool for VAWTs', Riva del Garda (Italy), September 6–7, 2018.
- [49] A. Bianchini, F. Balduzzi, L. Haack, S. Bigalli, B. Müller, and G. Ferrara, 'Development and Validation of a Hybrid Simulation Model for Darrieus Vertical-Axis Wind Turbines', presented at the ASME Turbo Expo 2019: Turbomachinery Technical Conference and Exposition, Nov. 2019. 10.1115/GT2019-91218.
- [50] P. Bachant, A. Goude, and M. Wosnik, 'Actuator line modeling of vertical-axis turbines', *ArXiv160501449 Phys.*, Sep. 2018, Accessed: Jun. 16, 2020. [Online]. Available: <http://arxiv.org/abs/1605.01449>.
- [51] 'Two-equation eddy-viscosity turbulence models for engineering applications | AIAA Journal'. <https://arc.aiaa.org/doi/abs/10.2514/3.12149?journalCode=aija> (accessed Jul. 06, 2022).
- [52] Balduzzi F, Bianchini A, Ferrara G, Ferrari L. Dimensionless numbers for the assessment of mesh and timestep requirements in CFD simulations of Darrieus wind turbines. *Energy* Feb. 2016;97:246–61. <https://doi.org/10.1016/j.energy.2015.12.111>.
- [53] 'A Comparison of Actuator Disk and Actuator Line Wind Turbine Models and Best Practices for Their Use', *Aerospace Sciences Meetings*. <https://arc.aiaa.org/doi/10.2514/6.2012-900> (accessed Jul. 04, 2022).



Contents lists available at ScienceDirect

Energy Conversion and Management

journal homepage: www.elsevier.com/locate/enconman

Is the Actuator line method able to reproduce the interaction between closely-spaced Darrieus rotors? a critical assessment on wind and hydrokinetic turbines

Omar Sherif Mohamed, Pier Francesco Melani, Francesco Papi, Francesco Balduzzi, Alessandro Bianchini*

Department of Industrial Engineering, Università degli Studi di Firenze, Via di Santa Marta 3, 50139, Firenze, Italy

ARTICLE INFO

Keywords:

Actuator Line Model (ALM)
VAWT
Darrieus
Hydrokinetic Turbines
Computational Fluid Dynamics (CFD)
Proper Orthogonal Decomposition (POD)

ABSTRACT

The deployment of multiple closely-spaced Darrieus turbines has recently gained interest in the academic and industrial sectors due to their potential to increase power output and wake recovery. However, the computational cost of accurate three-dimensional, blade-resolved CFD analyses rapidly becomes unfeasible as long as multiple rotors are added, putting emphasis on the need for a more practical, yet accurate, simulation method. Based on recent assessments, the Actuator Line Method (ALM) has been shown to represent the most interesting solution for Darrieus turbines' simulations. However, since the blade-flow interaction is not solved, no evidence was available to date on whether ALM can capture all the physical phenomena related to turbine mutual interaction.

In this study, the reliability and limitations of the ALM in simulating closely-spaced Darrieus turbines were assessed using two pairs of turbines with varying geometrical and operational characteristics, focusing on turbine loads, local flow field, and wake development. The results demonstrate that the ALM is effective for simulating twin-rotor Darrieus turbines, particularly at medium and high tip-speed ratios, providing satisfactory predictions of instantaneous blade loads and capturing the increase in torque due to the mutual interaction between adjacent rotors. The ALM is also able to accurately reproduce the flow field across the twin rotors, as demonstrated through Proper Orthogonal Decomposition (POD) analysis of the wake flow field. With an average simulation time of 7 h/rev/core, compared to 508 h/rev/core for blade-resolved CFD, this study provides evidence that the ALM is a low-cost and reliable tool for simulating closely spaced Darrieus turbines, thus representing the most suitable tool to develop new VAWT applications like floating wind turbines or hydrokinetic installations.

1. Introduction

1.1. Background and motivation

While the industry is still reluctant to invest in technologies for wind and tidal/hydrokinetic power deviating from the well-established three-blade horizontal-axis rotor, an increasing amount of research is pointing out that the vertical-axis concept has very interesting features for innovative installations. On the one hand, Vertical-Axis Wind Turbines (VAWTs) are very promising for offshore floating applications, both at the level of the isolated rotor, thanks to a lower center of gravity and therefore improved system dynamics [1], and at the wind farm one, as VAWTs can boost the farm energy harvesting by entraining momentum

from the upper layers of the atmosphere [2]. On the other hand, Vertical-Axis Hydrokinetic Turbines (VAHTs) already represent a low impact and cost-effective solution for the exploitation of the so-called "hidden hydropower", i.e., the one coming from channels and small rivers [3]. In spite of these advantages, further research is currently required to improve the energy conversion efficiency of these machines and fill their gap with respect to their horizontal-axis counterparts. In this perspective, one key topic that has been studied extensively in the last years are array effects in closely spaced Darrieus machines; in fact, it has been observed that a VAWT rotor has a higher efficiency in a row than when it is isolated. This effect was first shown Dabiri [4]. Following his experimental campaign, several studies were conducted, mainly using two-dimensional Reynolds-Averaged Navier-stokes (RANS) simulations, to investigate the physical mechanisms underlying such

* Corresponding author.

E-mail address: alessandro.bianchini@unifi.it (A. Bianchini).

<https://doi.org/10.1016/j.enconman.2023.117473>

Received 18 May 2023; Received in revised form 19 July 2023; Accepted 25 July 2023

Available online 29 July 2023

0196-8904/© 2023 Elsevier Ltd. All rights reserved.

Nomenclature			
<i>Latin symbols</i>		ϕ_n	Matrix of left eigenvectors [-]
A	Turbine area [m ²]	ϕ_V	Matrix of right eigenvectors [-]
C_L	Lift coefficient [-]	χ_n^m	Entries to the matrix of snapshots (dimensionless velocity fluctuations) [-]
C_p	Power coefficient [-]	Ω	Vorticity [s ⁻¹]
D	Turbine diameter [m]	ω	Rotor rotational speed [rad.s ⁻¹]
k	Turbulent kinetic energy [m ² .s ⁻²]	<i>Acronyms</i>	
L	Turbulent length scale [m]	AC	Aerodynamic Centre
P	Power [kW]	ALM	Actuator Line Method
R	Turbine radius [m]	AoA	Angle of Attack [deg]
Re_c	Chord-based Reynolds number [-]	BSC	Blade-Spoke Connection [-]
U	Matrix of snapshots [-]	CFD	Computational Fluid Dynamics
u_n	Left eigenvector [-]	CSS	Cubic Spline Smoothing
V_∞	Freestream velocity [m.s ⁻¹]	HAWT	Horizontal Axis Wind Turbine
V_m	Right eigenvector [-]	LES	Large Eddy Simulation
<i>Greek symbols</i>		POD	Proper Orthogonal Decomposition
β	Kernel width [m]	RANS	Reynolds-Averaged Navier-Stokes
Δh	Mesh element size [m]	SST	Shear Stress Transport
θ	Azimuth angle [deg]	SVD	Singular Value Decomposition
ρ	Flow density [kg.m ⁻³]	TSR	Tip Speed Ratio [-]
$\sum$	Matrix of eigenvalues [-]	VAHT	Vertical Axis Hydrokinetic Turbine
σ_m	Eigenvalue [-]	VAWT	Vertical Axis Wind Turbine

performance enhancement [5–7]. The favourable deviation in the absolute inflow direction and wake contraction happening in the gap between the rotors were identified as responsible for power augmentation. Other studies focused on optimising the layout of the turbines in terms of spacing and relative angle between the rotors [8–10]. They concluded that the maximum power output could be achieved via the *in-line* arrangement, i.e., zero oblique angle between the rotors, and spacing between 1.5 and 1.8D. The few three-dimensional studies available to date, for example the one of Nazari et al. [11], indicated two-dimensional studies as optimistic in determining torque production and wake dissipation rate. This kind of simulations, nonetheless, is at the current state-of-the-art not feasible at a research and industrial level, due to prohibitive computational power requirements.

Therefore, the attention of the scientific community has been directed towards the development of low- or medium-order approaches, able to capture the main effect on performance and wake recovery of blade-resolved CFD, but at a notably lower computational cost. Low-order tools were proposed by Araya et al. [12], who developed an analytical model based on the Rankine body formulation, and Nishino et al. [13], who proposed a theoretical model to estimate the performance of tidal Darrieus turbine arrays within a finite-width channel. Although considered beneficial for the rough estimation of the energy output of a Darrieus turbine cluster, these models cannot capture the complex three-dimensional flow structures that occur either in the wake or in the mutual interaction region between the rotors. Tavernier et al. [14] employed a two-dimensional vortex model to study a pair of Darrieus wind turbines, while Gauvin-Tremblay and Dumas [15] developed the Effective Performance Turbine Model to investigate the local blockage effect in an array of VAHTs.

To have an insight on these aspects, CFD must therefore be reintroduced. This led to the development of the so-called “hybrid” methods that combine CFD to solve the flow field and a lumped-parameter aerodynamic model, usually based on input polar data, to compute the rotor forces, which are inserted in the computational grid as volume forces [16].

Among the hybrid simulation tools available, the Actuator Line Method (ALM) stands out for its accuracy and computational efficiency. Several studies [17–19], including contributions from the authors

[20–22], have demonstrated the capability of this approach to simulate isolated Darrieus rotors. However, there are only a few studies in the literature that employ the ALM for multiple rotors. For example, Hezaveh et al. [23] used the ALM in conjunction with a Large Eddy Simulation (LES) model to evaluate the feasibility and performance of a VAWTs wind farm. While Raj used the ALM to investigate the spacings between two closely spaced rotors [24]. In a recent work by Zhang et al. [25], the ALM was coupled with RANS simulations to investigate the synergy effect of two and three VAWTs. It is worth noting that these latter studies were primarily focused on optimizing the cluster synergy effect or studying the wake on a farm scale, without critically investigating the ability of the ALM to capture the flow interactions that occur between closely deployed Darrieus rotors.

It is important to mention that the authors have extensively developed the ALM formulation utilized in this work by incorporating additional models tailored for VAWT rotors. This includes evaluating dynamic stall model settings [26], incorporating ad hoc models to account for blade-spoke connections and pitching moment [20], and verifying the use of the *LineAverage* method to sample the Angle of Attack (AoA) in VAWTs [27,28]. Furthermore, a dedicated sensitivity analysis of this ALM formulation has been conducted recently to investigate the effects of kernel shape and size, as well as spatial sensitivity analysis to loads [22].

1.2. Novelty and contribution of the study

Moving from this background, the present study aims at understanding if a hybrid simulation model like the ALM, which cannot intrinsically capture higher-order fluid-dynamic phenomena like blade-vortices interaction, is anyhow able to cope with the complex array effects occurring when multiple rotors are deployed in close proximity, i.e., the suppression of the streamtube expansion effect in the gap between the rotors, together with the associated performance augmentation. These effects were extensively analysed using two-dimensional blade-resolved CFD in a recent study by the authors [29]. Proving that a computationally efficient method like ALM is suitable to this end is of particular industrial relevance, as it can become a new industry standard to promote the emerging technology of hydrokinetic turbines deployed

Table 1
Operational and geometrical characteristics of the turbine pairs used in the present work.

Variable	3B-VAWT	2B-VAHT
Airfoil shape	DU06W200	DU06W200 (reversed)
Number of blades [#]	3	2
Diameter [m]	1.2	2
Chord [m]	0.128	0.25
Solidity (nc/D) [-]	0.32	0.25
Tip Speed Ratio (TSR) [-]	2.7 – 3.2	2.5—3
Centre-to-centre spacing [-]	1.5D	2.5D
Freestream velocity [m/s]	8	2

in arrays. In particular, this represents the basic theoretical step needed to build upon when moving to three-dimensional simulations of real machines. To date, indeed, no prove that ALM could capture twin VAWT interactions was available.

The ALM tool developed over the years specifically for the simulation of vertical-axis turbines [20,22] has been employed. The latter features a novel *LineAverage* (*LineAvg*) method for accurate AoA sampling, as well as advanced aerodynamic sub-models to account for phenomena typically associated with Darrieus turbines, such as dynamic stall, pitching moment deriving from the selected blade-spoke connection, and flow curvature. ALM simulations were performed for both a wind and an hydrokinetic rotor, and validated against the blade-resolved CFD data from Zanforlin et al. [5] and from a recent work of the authors [29], respectively. The comparison of the two datasets was carried out in terms of:

- turbine performance*, in terms of overall performance and torque profiles over a revolution;
- flow kinematics*, with particular reference to the local AoA and velocity magnitude, which were sampled from both ALM and blade-resolved simulations via the *LineAverage* sampling approach [30], recently validated by the authors for the use on vertical-axis turbines [16];
- wake development*. One of the elements of novelty of the present study is the use of Proper Orthogonal Decomposition (POD) to analyze the capability of the ALM to reproduce the wake of closely-spaced Darrieus turbines. Such information is of capital relevance for many potential applications, including high-power-density wind farms based on VAWTs [31] or arrays of hydrokinetic turbines to be installed in channels or rivers [32].

1.3. Structure of the study

The structure of this study is as follows. Section 2 provides details on the geometrical and operational characteristics of the selected test cases. In Section 3, the characteristics of the different numerical methodologies used herein are presented, including a brief description of the ALM base formulation, CFD set-up, and the adopted POD model. The results are presented in Section 4, where comparisons between the blade-resolved RANS results and the ALM are presented in terms of power coefficient, local flow kinematics, and wake flow field. Section 5 discusses the results, highlighting their relevance to go beyond the current state-of-the-art. Finally, Section 6 provides the conclusions and outlines potential future work.

2. Test cases

In the present study, two different configurations of twin Darrieus rotors were used as case studies. More specifically, analyses refer to:

- 3B-VAWT - WindSpire**: a 3-blade Darrieus VAWT that has been extensively tested for different configurations of closely-spaced rotors. Pioneering field measurements were reported by Dabiri et al.

[4], in addition to the two-dimensional blade-resolved RANS results reported by Zanforlin et al. [5], and the results of Hezaveh et al. [23], which have been produced using a different formulation of the ALM. In the present study, only the two dimensional blade-resolved CFD results of [5] have been used as benchmarking data to validate the current ALM formulation;

- 2B-VAHT - HE-PowerGreen**: an industrial prototype of 2-blade Darrieus VAHT early presented in [32], for which multiple design configurations have been investigated in previous works [3,29]. It has been also studied in [21], where an investigation on a possible active pitch control strategy has been conducted using ALM. In this study, a set of dedicated twin-rotors, blade-resolved RANS simulations have been carried out to be used for the ALM validation.

The corresponding geometrical and operational characteristics for the two rotors are reported in Table 1. The selection of these test cases was justified by the quantity and quality of benchmark data available, as well as the corresponding variety of rotor solidities, working fluid (air vs. water), blade Reynolds number ranges. In this perspective, the validation is thought to be as general as possible. It is worth noticing that, while having different solidity, both test cases fall in the medium solidity range, which is particularly indicated for hybrid simulation models, as their accuracy in fact strongly decreases going toward high-solidity configurations [33].

The operating conditions of interest were selected so that the ALM tool could be assessed in different modes of operation. For the 3B-VAWT rotor, two operating points were picked, the optimum TSR = 2.7, where the dynamic stall is relevant, and TSR = 3.2, where blockage effects are dominant. On the other hand, for the 2B-VAHT rotor, TSRs of 2.5 and 3 were selected. TSR = 3 is the optimum operating point for twin rotors as concluded in [29], while at TSR = 2.5, a significant impact of dynamic stall is apparent, and has been successfully modelled for a stand-alone configuration in [2018].

3. Methodology

3.1. Actuator Line Method (ALM)

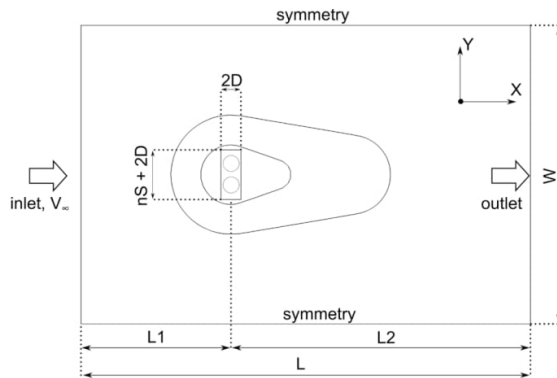
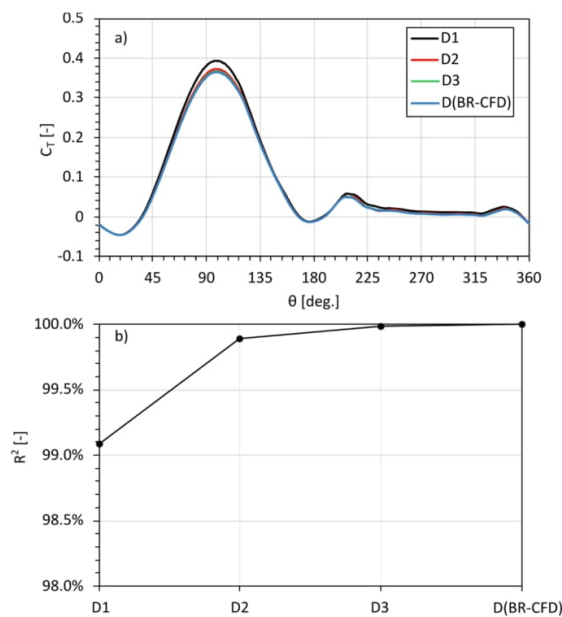
The Actuator Line Method (ALM) [16] is a computational tool that is being extensively used for simulating wind turbines. The corresponding workflow is the following. First, the position of the actuator line corresponding to the turbine blade is located on the computational grid. Then, the blade Reynolds number and angle of attack are sampled from the local flow field and are combined with the available polar data and sub-models to compute the corresponding aerodynamic forces. The latter are eventually projected into the computational domain as momentum source terms in the Reynolds-Averaged Navier-Stokes (RANS) solver. In the present work, the ALM tool developed over the years by the authors and tailored to the simulation of vertical-axis turbines [20] has been adopted and set-up as following:

- Sampling of the local flow field**: the *LineAverage* method is used to estimate the blade angle of attack (AoA) from the local flow field. Originally proposed for horizontal-axis wind turbines by Jost et al. [30] and validated by the authors for the application to vertical-axis turbines [27,28], this method calculates the undisturbed velocity as an integral average of the velocity field along a closed line around the airfoil. In the present study in particular, a circular line with a radius of $r = 1c$, $N = 80$ equally spaced points and centered on the airfoil quarter-chord is used. Moreover, to reduce the background noise, a low-pass filtering is applied to the sampled data using a Cubic Spline Smoothing (CSS) algorithm [34], according to the procedure outlined in [26];
- Pitching moment and BSC corrections**: to account for two phenomena specific to vertical-axis turbines, ad hoc corrections are applied to the predicted torque. First, since the pitching moment of the airfoil acts

Table 2

Test cases for the sensitivity analysis on domain dimensions.

Case	D1	D2	D3	D(BR-CFD)
L1/D	10	15	20	20
L/D = (L1 + L2)/D	30	45	60	80
W/D	20	30	40	42
Aspect ratio (L/W)	1.5			1.9

**Fig. 1.** Computational domain and boundary conditions.**Fig. 2.** Results of domain independence test for 2B-VAHT pair at TSR = 3: a) instantaneous torque coefficient and b) coefficient of determination.

in the blade rotational plane, an additional contribution is included in the shaft torque computation [35]. Second, in the cases where the Blade-Spoke Connection (BSC) point of the blade support strut does not coincide with the blade's AC, the AoA and load distribution must be modified accordingly [18,19]. It is worth noting that, for the selected test case, the blade is mounted at the quarter-chord and thus the latter correction has no effect (see Table 2);

- **Polar data:** the accuracy of the ALM code in predicting the performance of a rotor depends on the quality of the polar data used. This is particularly important for small Darrieus rotors, such as the ones considered in the present study, where the Reynolds numbers are relatively low, and the impact of laminar separation bubbles on the aerodynamic performance of the airfoil is more significant. In addition, one should remember that, as any other method that makes use of tabulated data to model blade aerodynamics, the use of data referring to the virtually transformed airfoil deriving from virtual camber correction are mandatory [36]. In the present case, pre-stall aerodynamic coefficients for both rotors were determined using the viscous panel method software XFOIL [37], which proved to be robust when benchmarked against more complex methods. Additionally, both sets of data were extended over the deep stall region, up to an angle of attack of $\pm 180^\circ$, using the Viterna-Corrigan model [38], as recommended by Bianchini et al. [39] for Darrieus turbines applications;
- **Berg dynamic stall model:** the present code incorporates the dynamic stall model proposed by Berg [40] to accurately capture the impact of unsteady flow on the blade loads, as explained in [26]. This model introduces a delay in the measured angle of attack proportional to the flow reduced frequency to embed dynamic effects in the solution. The lift and drag coefficients are obtained as a linear interpolation between the static and dynamic coefficients, weighted over the constant A_M . The A_M values were fine-tuned to be 6 and 20 for 3B-VAWT and 2B-VAHT twin rotors, respectively, following the procedure described in [26]. To determine the stall angle, which is a crucial step in determining the slope of the lift curve in the pre-stall region, two methods were evaluated in the previous work discussed in [26]. The first method, *MaxLift*, selects the static stall angle as the point of maximum lift in the pre-stall region, following the same approach used to extend the polar data in the post-stall region. The second method, *LinReg*, selects the static stall angle as the angle giving the line that best approximates the lift curve in the pre-stall region. The former method is straightforward to implement when the operating Reynolds number is high enough to keep the lift behavior nearly linear during the pre-stall region. In contrast to the *MaxLift* which can be beneficial when the operating chord-based Reynolds number is low, and the pre-stall lift behavior is nonlinear due to the presence of a viscous separation bubble. Considering the range of the operating Re_c for the current case studies, for 3B-VAWT rotors, *MaxLift* was used, while *LinReg* was implemented for the 2B-VAHT twin rotors.

Additional information about the code formulation can be found in [20].

Regarding the *Frozen ALM* approach, this was originally introduced by [41], and has been recently used by the authors to study the impact of the Regularization Kernel function for a single Darrieus rotor [22]. This approach, while not specifically thought for design (since aerodynamic coefficients data coming from blade-resolved CFD are needed) is of particular use whenever one needs to differentiate the errors stemming from the aerodynamic coefficients or dynamic stall model and those resulting from other factors such as the projection function, angle of attack (AoA) sampling, spatial or temporal discretization. In this study, the *Frozen ALM* is employed in the wake analysis section to effectively identify the sources of discrepancies between the ALM wake and that of the blade-resolved CFD.

3.2. CFD setup

Two-dimensional simulations were performed using the unsteady Reynolds-Averaged Navier-Stokes (URANS) method, employing the commercial software ANSYS® FLUENT® in its 2020 R2 release. The selected turbulence closure model was the $k-\omega$ Shear Stress Transport (SST) model, renowned for its accuracy and stability in unsteady flow

Table 3

Tested grids for the ALM region mesh sensitivity analysis.

Case	Δh_{ALM}	No. of elements
G1.1	β	83,538
G1.2	$\beta/1.5$	180,639
G1.3	$\beta/2$	313,777

Table 4

Tested grids for the wake zone mesh sensitivity analysis.

Case	$\Delta h_{NearWake}$	No. of elements
G2.1	$3\Delta h_{ALM}$	104,258
G2.2	$2.5\Delta h_{ALM}$	131,277
G2.3	$2\Delta h_{ALM}$	180,639
G2.4	$1.5\Delta h_{ALM}$	285,777

conditions. These conditions frequently arise in VAWT blades, where significant flow separations and pressure gradients occur [42]. It is worth noting that the y^+ value for all blade-resolved CFD cases remained below 1, satisfying the prerequisites of the SST model.

Furthermore, other simulation details, including the domain, grid, and time stepping sizes, were consistent for both the blade-resolved CFD and the ALM. The pressure-based solution was employed using a coupled algorithm, and a second-order upwind scheme was utilized for the spatial discretization of all transport equations. The time differencing employed a bounded second-order implicit scheme. Additionally, the residuals for all variables were set to 10^{-5} , and 30 inner iterations were performed per time step. These simulation settings were chosen based on comprehensive sensitivity analyses conducted by some of the authors, as referenced in [43,44].

For a detailed description of the blade-resolved CFD setup, please refer to the authors' latest publication in [29]. The CFD domain, boundary conditions, and grid details for the ALM are reported and discussed in the subsequent sub-sections.

3.2.1. Domain and boundary conditions

The computational domain was an open field, with its dimensions parametrized with respect to the turbine diameter D as shown in Fig. 1. The partitioning and sizes of the wake zones were established following the approach outlined in [45]. This methodology aimed to minimize flow disturbances caused by the turbine, thereby justifying the use of standard far-field boundary conditions commonly employed in external flow simulations.

To ensure the numerical independence of the solution from the domain size, a domain sensitivity analysis was performed. Different test cases with varying domain sizes were evaluated for the 2B-VAHT, as reported in Table 2, which also includes the domain size of the blade-resolved CFD evaluated on an ALM case. Fig. 2 presents the results of the domain sensitivity assessment in terms of the instantaneous torque coefficient, C_T , and coefficient of determination, R^2 . Some of the authors have introduced R^2 as a criterion for evaluating the spatial or temporal numerical independence, as described in [44]. This criterion considers both the instantaneous and average values of the errors. In this case, R^2 was calculated as follows:

$$R^2 = 1 - \frac{\sum_{\theta=0^\circ}^{360^\circ} [C_T(\theta) - C_{T,final}(\theta)]^2}{\sum_{\theta=0^\circ}^{360^\circ} [C_T(\theta) - C_{T,avg}]^2} \quad (1)$$

where $C_T(\theta)$ represents the instantaneous torque coefficient of a given case, $C_{T,final}(\theta)$ denotes the instantaneous torque coefficient of the largest domain case, D(BR-CFD) in the present analysis, and $C_{T,avg}$ represents the average torque over a full revolution. From the results shown in Fig. 2, no significant differences can be observed between D02, D03, and D(BR-CFD). Notably, the R^2 value for D02 is above 99.9%. Based on these findings, the domain dimensions of D02 were selected for

the subsequent simulations.

Regarding the boundary conditions, uniform values were prescribed at the inlet for velocity (8 m/s for the 3B-VAWT and 2 m/s for the 2B-VAHT), turbulent kinetic energy (k), and length scale (L), tailored to match the testing conditions of the specific test case. The turbulent quantities were estimated based on the inlet turbulent intensity and length scale. For the 3B-VAWT, a turbulent intensity of 4% with a length scale of 1 m was set, while for the 2B-VAHT, a turbulent intensity of 0.25% with a length scale of 2.84 m was used. As for the outlet boundary, a uniform atmospheric pressure was specified, and a symmetry condition was enforced on the other boundaries.

3.2.2. Computational grid

A hybrid meshing strategy was employed to discretize the CFD domain into three separate fluid zones based on the main features commonly found in a Darrieus turbine operating in an *open-field* environment. The first zone is the rotor ALM region, which encompasses the turbine rotor and is discretized with a structured mesh to fulfil the requirements of the ALM. The second zone is the wake region, which requires an unstructured quad-dominant mesh to capture the turbine blockage and wake development. This region is divided into a near-wake and a far-wake zone. The near wake region is extended 6D downstream, while the far wake region is 16D downstream.

In a previous study [22], a grid independence test was performed by the authors to determine the optimal mesh resolution in the ALM region based on turbine loads. The conclusion drawn from that study was that $\Delta h_{ALM} = \beta/1.5$ provides a favourable balance between accuracy and computational time, where β represents the kernel width ($\beta = 0.25c$). However, in the current analysis, two separate grid independence tests were conducted, focusing on both the ALM and wake regions.

The first grid sensitivity assessment aimed to ensure that load estimation remains independent of the grid size, and to validate the conclusions reached from the single rotor analysis for the twin rotor analysis as well. In this test, different cases with varying mesh sizes are presented in Table 3, where different element sizes were implemented in the ALM region. The meshing strategy in this assessment follows that of the single rotor analysis reported in previous studies. Specifically, the near wake region is always taken as double the size of the ALM region ($\Delta h_{NearWake} = 2\Delta h_{ALM}$), while the far wake region is double the size of the near wake region ($\Delta h_{FarWake} = 2\Delta h_{NearWake}$).

The second grid sensitivity analysis focused on the wake regions, where the element size in the ALM region remained constant while different grid sizes were applied to the near wake region, $\Delta h_{NearWake}$. The element size in the far wake region was consistently set at twice the size of the near wake region across all tested meshes. The specific details of the evaluated meshes can be found in Table 4.

Fig. 3 presents the results of the mesh independence tests. On the top left side of the figure, the results of the first grid assessment are shown in terms of the instantaneous torque coefficient, it is important to note that here, all the cases reached an $R^2 > 99.9\%$. However, increasing the grid size in the ALM region above $\beta/1.5$ lead to volatility in the calculated load as highlighted at $75^\circ < \theta < 120^\circ$, as observed also before for the single rotor [22]. Therefore, the results of the first grid independence test led to the conclusion that $\Delta h_{ALM} = \beta/1.5$ is the optimal mesh size. Thus, G1.2 was selected for the subsequent grid independency test. On the right side of Fig. 2, the results of the wake grid sensitivity analysis are illustrated in terms of velocity profiles at X/D of 1 and 3. The solid and dashed lines represent the coefficient of determination, R^2 , for the velocity profiles. In this test, R^2 was calculated as:

$$R^2 = 1 - \frac{\sum_{Y/D=-2.5}^{2.5} [V(Y/D) - V_{final}(Y/D)]^2}{\sum_{Y/D=-2.5}^{2.5} [V(Y/D) - V_{avg}]^2} \quad (2)$$

where $V(Y/D)$ represents the local velocity value at a specific Y/D for the given mesh, $V_{final}(X/D)$ indicates the local velocity value of the finest mesh (G2.4 in this case), and V_{avg} denotes the average velocity along the

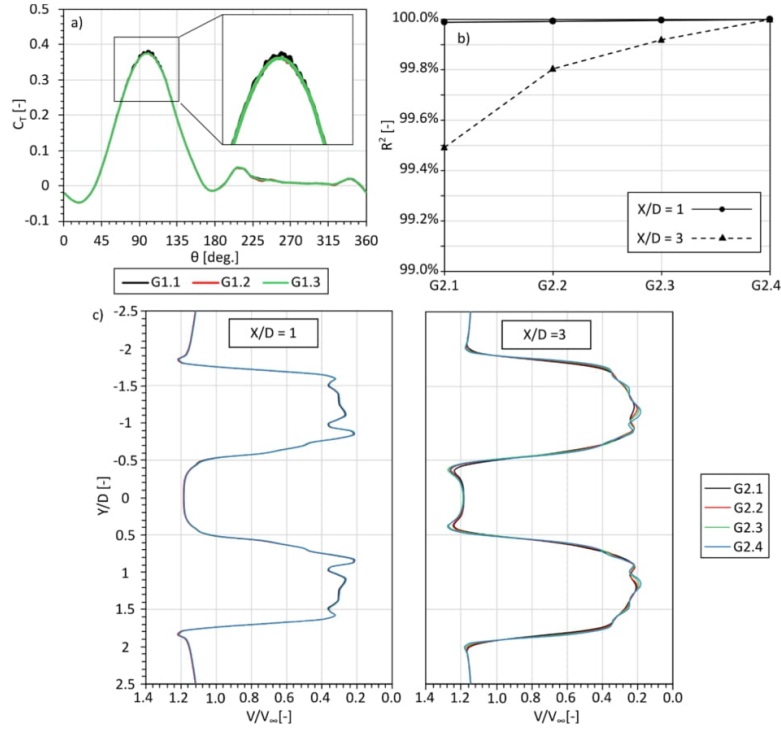


Fig. 3. Comparison between the different grid resolutions in terms of a) torque coefficient, b) coefficient of determination, and c) velocity profiles for the 2B-VAHT pair at TSR = 3 at $X/D = 1$ and 3.

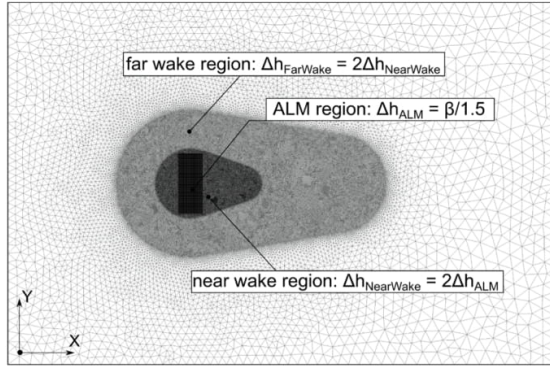


Fig. 4. Details of the computational grid.

Table 5

A comparison of the numerical settings between the blade-resolved CFD and ALM simulations.

Numerical parameter	Blade-resolved CFD	ALM
Grid number of elements	1,016,848	180,639
Domain size	42D × 90D	30D × 45D
Angular step size ($\Delta\theta$)	0.4°	1°

given mesh. At $X/D = 1$, all grids exhibit identical velocity profiles. However, at $X/D = 3$, slight variations can be observed, with G2.3 displaying an R^2 value exceeding 99.9%. Based on these findings, G2.3 was chosen for subsequent simulations. Fig. 4 illustrates the final selected mesh, including the representation of the mesh zones. Additionally, Table 5 presents a comparison between the blade-resolved CFD model and the ALM in terms of spatial and temporal discretization.

3.3. Proper Orthogonal Decomposition (POD)

POD is a powerful tool for analysing transient flow fields due to its ability to identify the main coherent spatial structures, or modes, contributing to the flow behaviour. Each mode represents a distinct spatial pattern, with a corresponding temporal coefficient that describes its time evolution. The modes are ordered based on their energy levels, with the first few modes typically capturing most of the flow dynamics [46]. POD has been recently employed to study the wake behind wind turbines. For example, Chen et al. [47] utilized RANS turbulence models to investigate the effect of gurney flaps on the performance of an H-shaped Darrieus VAWT. More recently, Sheidani et al. [48] used the POD to compare between RANS and LES simulations for a Darrieus hydrokinetic turbine.

In the present work, POD was applied to both simulations, in order to compare the wakes produced by the blade-resolved CFD and the ALM. More in detail, the flow field data at each spatial point on the two-dimensional domain was stored into a unit vector, χ_n^m , where n is the number of spatial points available, and m is the number of timesteps. Each value of χ_n^m is the fluctuating component of the velocity magnitude, obtained by removing the mean value from the instantaneous one. In this way, an $n \times m$ matrix U could be formed as:

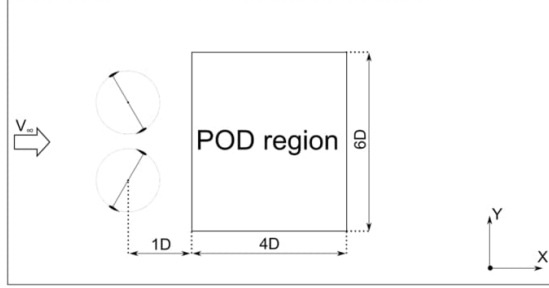


Fig. 5. POD analysis region.

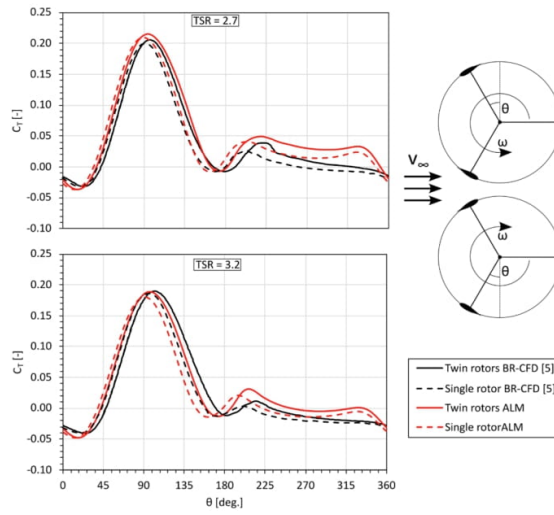


Fig. 6. Comparison between blade-resolved RANS [5] and ALM in terms of torque coefficient for 3B-VAWT twin rotors in the inward configuration at TSRs = 2.7 and 3.2.

$$U = \begin{bmatrix} \chi_1^1 & \dots & \chi_1^m \\ \vdots & & \vdots \\ \chi_n^1 & \dots & \chi_n^m \end{bmatrix} \quad (1)$$

This matrix is then decomposed via Singular Value Decomposition (SVD) into three other matrices, two of which contain the coherent structures in the spatial and temporal domains, in addition to the matrix of eigenvalues:

$$U = \phi_u \Sigma \phi_v^T = \begin{bmatrix} \vdots & & \vdots \\ u_1 & \dots & u_n \\ \vdots & & \vdots \end{bmatrix} \begin{bmatrix} \sigma_1 & 0 & 0 \\ 0 & \ddots & 0 \\ 0 & 0 & \sigma_m \end{bmatrix} \begin{bmatrix} \dots & V_1^T & \dots \\ \dots & V_2^T & \dots \\ \dots & V_m^T & \dots \end{bmatrix} \quad (2)$$

where ϕ_u is the left eigenvectors matrix that contains the spatial coherent structures in the n points, while ϕ_v^T is the right eigenvectors matrix, physically representing the evolution of each mode in the m points. Σ , on the other hand, is a diagonal matrix of singular values, σ , organized in descending order, allowing the ordering of spatial modes in ϕ_u based on the corresponding energy level. As most of its energy is usually contained in the first few modes. The singular values are the square roots of the eigenvalues of the covariance matrix of the original data matrix, U . The percentage of energy corresponding to each mode is then computed as the square of the corresponding singular value divided

by the sum of the squares of all singular values. From the statistical point of view, this quantity is equivalent to the total variance of the data explained by each mode. From the physical one, it corresponds to the total kinetic energy generated due to the velocity fluctuations in each mode.

The use of POD allowed to identify the core differences between the wakes predicted by ALM and blade-resolved CFD, thus providing a deeper understanding of how each formulation influences the wake development. In this study, the POD analysis was conducted within a selected area spanning 6D in the crosswind direction and 5D in the streamwise direction, located 1D downwind from the centre of the rotors as indicated in Fig. 5. Instantaneous velocity magnitude data were stored for the three cycles after the 50th revolution to ensure that the flow field had reached a steady-state periodic condition past the turbines. It should be noted that the POD analysis was only carried out on the 2B-VAHT rotor due to the availability of detailed flow fields from blade-resolved simulations.

4. Results

The results are reported in this section for both the 3B-VAWT (section 4.1) and the 2B-VAHT (section 4.2) twin rotors. In each section, the load results are firstly compared to those of blade-resolved CFD in terms of power and torque coefficients (C_p and C_T , respectively). The C_p is defined as the ratio of power, P , extracted by the turbine to the power available in the wind, while the C_T is the dimensionless torque on the blades:

$$C_p = \frac{P}{0.5\rho A V_\infty^3} \quad (4)$$

$$C_T = \frac{T}{0.5\rho A R V_\infty^2} = \frac{C_p}{TSR} \quad (5)$$

Secondly, the results of the local flow kinematics are reported and compared to that of the blade-resolved CFD. For the 3B-VAWT, the local velocity profile was sampled at a certain azimuth position and compared to the data published by Ref. [5]. While for the 2B-VAHT, this assessment was conducted by comparing the instantaneous angle of attack results. Finally, the wake flow field was investigated for only the 2B-VAHT due to the data availability. This was carried out by initially reporting the velocity profiles at different downwind distances, followed by a POD analysis where the coherent spatial modes resulting from both the ALM and blade-resolved CFD simulations are compared.

4.1. Three-blade VAWT: 3B-VAWT

This section presents the simulation results of the c , a three-bladed VAWT with medium solidity, to evaluate the ALM capabilities in simulating the performance of twin-rotor VAWTs. The results of this rotor pair include both inward and outward modes of rotation, with a distance of 1.5D between the two rotors. The operating range of this turbine is limited to relatively low chord-based Reynolds numbers, ranging from 130 k to 290 k within the assessed TSRs of 2.7 and 3.2.

4.1.1. Blade loading

For the *inward* configuration, the blade loading of the 3B-VAWT turbine at TSRs of 2.7 and 3.2 were analysed using the ALM. Fig. 6 shows the profile of blade torque coefficient (C_T) along one revolution, averaged over the two rotors. The results obtained from ALM were compared with the corresponding blade-resolved RANS simulations of Zanforlin et al. [5]. Additionally, the single rotor results, including those obtained from the single rotor ALM, were reported to provide a basis for comparison.

At TSR = 2.7, the results showed that the ALM was able to reconstruct the trend of the torque coefficient profile in the *upwind* region ($0^\circ \leq \theta \leq 180^\circ$), capturing the azimuthal position of the peak torque with a

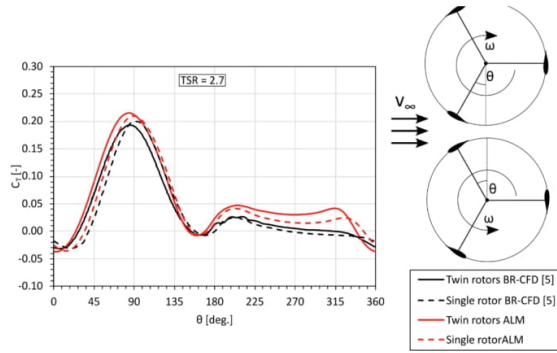


Fig. 7. Comparison between blade-resolved RANS [5] and ALM in terms of torque coefficient for 3B-VAWT twin rotors in the outward configuration at $TSR = 2.7$.

slight overestimation in its magnitude compared to the reference data. In the *downwind* section ($180^\circ \leq \theta \leq 360^\circ$), nonetheless, the ALM achieved a fair prediction of the torque coefficient trend, although overestimating energy extraction along the whole blade path between $\theta = 180^\circ$ and $\theta = 360^\circ$. At $TSR = 3.2$, in the *upwind* half, the trend of the torque coefficient profile matched that of the blade-resolved CFD, although with a slight lead of approximately 5° . Unlike at $TSR = 2.7$, the ALM provided a satisfactory estimation of the *downwind* torque, although still slightly overestimated with respect to the reference data.

Fig. 7 presents the same comparison for the outward configuration. The plot shows that during the first quarter of the azimuth angle range, where the interactions between adjacent blades are most significant ($0^\circ \leq \theta \leq 90^\circ$), the ALM torque coefficient trend showed a slight overestimation that increased gradually until reaching the peak torque near $\theta \approx 90^\circ$, where the ALM was able to capture its azimuthal position but with an overestimated magnitude. Then the gap between the ALM torque and that of the reference data decreased gradually until matching at $\theta \approx 170^\circ$. In the *downwind* azimuthal path on the other hand, torque production was overestimated along most of the path, as seen in the *inward* configuration. It is worth noticing that unlike in the *inward* configuration the ALM in the *outward* configuration accurately captured

the trend of the torque coefficient without any shifts in the azimuth position.

4.1.2. Velocity profiles

In order to evaluate the ALM ability to reproduce the flow field around the twin rotors, the velocity profiles in the streamwise (X) and crosswise (Y) directions were compared between the ALM and the blade-resolved CFD from Zanforlin et al. [5]. For comparison, the velocity profile of the clockwise rotor was reoriented to the direction of the anticlockwise rotor, and an average was calculated for both rotors.

Fig. 8 shows the velocity profiles sampled at $TSRs = 2.7$ and 3.2 upstream of the twin rotor. At $TSR = 2.7$, it can be seen that the ALM shows good agreement with the blade-resolved CFD data, with some discrepancies in terms of: (i) *upwind* blockage effect, in particular the X -velocity deficit at the rotor centre ($Y/D = 0$); (ii) slope of variation of the Y -velocity, i.e., degree of expansion of the flow at the sides of the twin rotor and the gap. The same considerations can be made for $TSR = 3.2$, but here the blockage effect induced by the twin rotor on the incoming flow is even more underpredicted with respect to blade-resolved CFD, both at

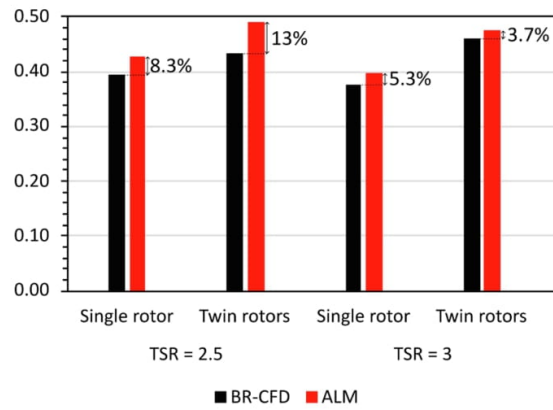


Fig. 9. Comparison between blade-resolved CFD and ALM in terms of average power coefficient 2B-VAWT single and twin rotors at $TSRs = 2.5$ and 3 .

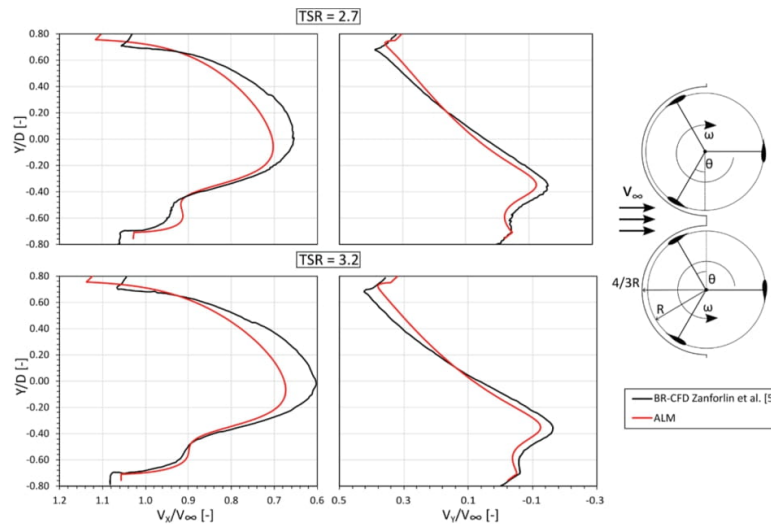


Fig. 8. Comparison between blade-resolved RANS [5] and ALM in terms of upstream velocity profiles for 3B-VAWT twin rotors at $TSRs = 2.7$ and 3.2 .

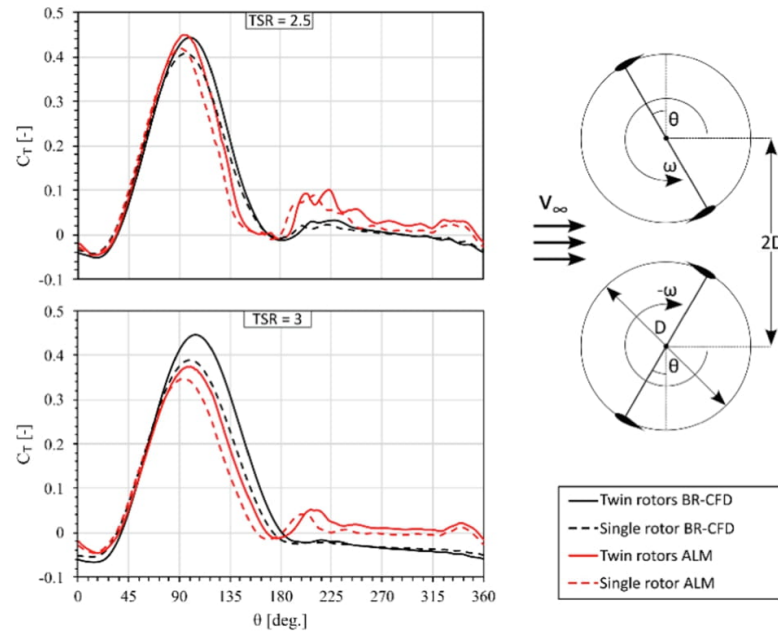


Fig. 10. Comparison between blade-resolved CFD and ALM in terms of instantaneous torque coefficient for 2B-VAHT single and twin rotors at TSRs 2.5 and 3.

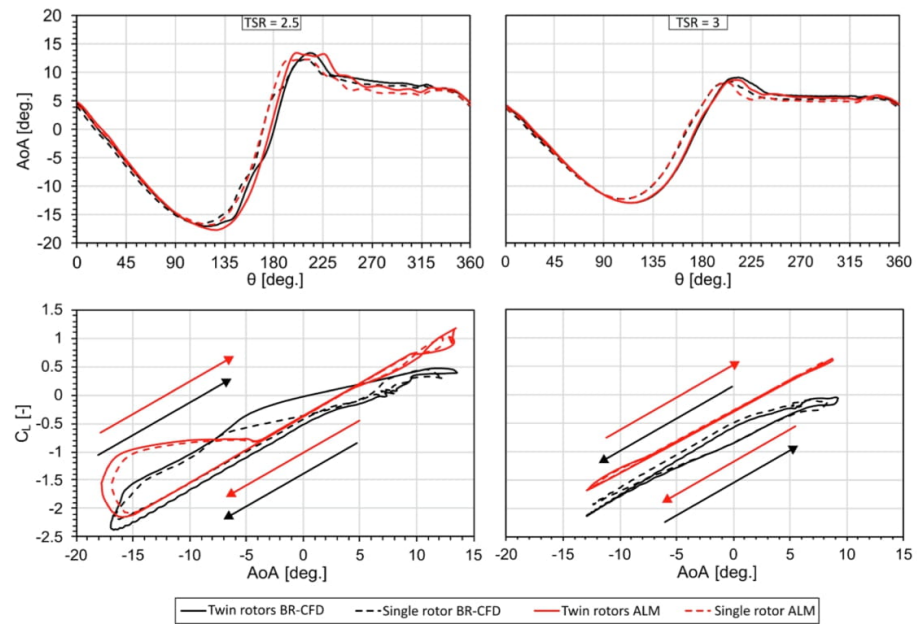


Fig. 11. Comparison between blade-resolved CFD and ALM in terms of AoA and lift hysteresis loop for 2B-VAHT single and twin rotors at TSRs = 2.5 and 3.

the sides and in the gap.

Based on previous considerations, it can be concluded that for the 3B-VAWT *WindSpire* test case, the ALM is able not only to fairly reproduce the flow pattern induced by the pairing of multiple Darrieus rotors, which, according to Zanforlin et al. [5], consists in the suppression of the flow expansion in the mutual interaction area ($Y/D < 0$) of the rotors,

but also to capture the corresponding enhancement of the torque production in the same region, with the following power augmentation effect. Nonetheless, two sources of discrepancy with respect to blade-resolved data can still be identified: (i) the low Reynolds numbers experienced by the blades (i.e., $Re_c < 300$ k), which make the assessment of the input polar data extremely critical (see Section 3); (ii) the

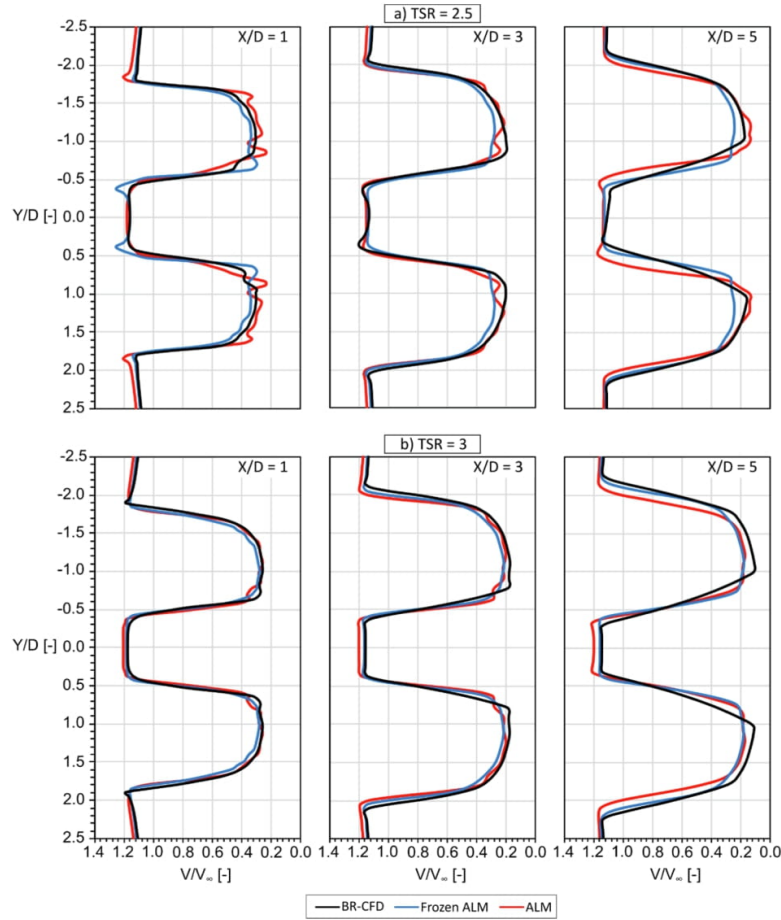


Fig. 12. Comparison between blade-resolved CFD and ALM at $\theta = 0^\circ$ in terms of wake velocity profiles sampled at $X/D = 1D, 3D$, and $5D$ for *2B-VAHT* twin rotors at TSRs = a) 2.5 and b) 3 at $\theta = 0^\circ$.

unsteady effects experienced by the blades could not be fully reproduced by the ALM whether through the use of Berg model or even without employing any dynamic stall model. The latter is more evident from the overestimation of the torque coefficient profile during the downwind azimuth positions observed for both inward and outward configurations and different TSRs. This observation aligns with a previous study for a stand-alone rotor [22].

4.2. Two-blade hydrokinetic turbine

In this section, the ALM results for the *2B-VAHT* inward counter-rotating twin rotors (see Section 2) are presented and compared to the blade-resolved CFD simulations from [29] for two different TSRs (2.5 and 3). In contrast with the *3B-VAWT* case, complete flow data were available for this turbine, enabling the advanced post-processing of the rotor flow via *LineAverage* sampling of the blade angle of attack, useful for a benchmarking of the ALM in terms of aerodynamic coefficients, and POD analysis of the wake.

4.2.1. Blade loading

Fig. 9 shows the cyclic average power coefficient C_P computed using blade-resolved CFD and ALM for both stand-alone rotor and twin rotors of *2B-VAHT* at TSRs 2.5 and 3. In general, it can be observed that the

ALM overestimates the average C_P in all cases. This overestimation is less prominent at $TSR = 3$ compared to $TSR = 2.5$.

However, the average value could hide significant differences in the instantaneous torque values, which could have been compensated during the averaging process along one rotor revolution. To take that into consideration, Fig. 10 reports the comparison between ALM and CFD in terms of instantaneous C_T along one rotor revolution, for both the stand-alone and the twin rotor. At $TSR = 2.5$, the ALM was able to reproduce the torque profile in the whole *pitch-up* phase ($0^\circ < \theta < 90^\circ$), up to its peak value for both single and twin rotors. In the *pitch-down* phase ($90^\circ < \theta < 180^\circ$) instead, the ALM tends to underpredict torque production in both stand-alone and twin configurations. The opposite trend is observed in the *downwind* region ($180^\circ < \theta < 360^\circ$), where the ALM overpredicts the blade loads with respect to reference data along the whole blade path. This behaviour is even more accentuated at $TSR = 3$, where, for both stand-alone and twin configurations, the ALM also anticipates the position of the torque peak of ca. 10° .

4.2.2. Blade kinematics

In order to gain a better insight into the discrepancies in terms of loads observed between ALM and blade-resolved CFD in Section 4.2.1, Fig. 11 presents a similar comparison, but focusing on the local instantaneous angle of attack α and lift coefficient C_L . It is worth noting that

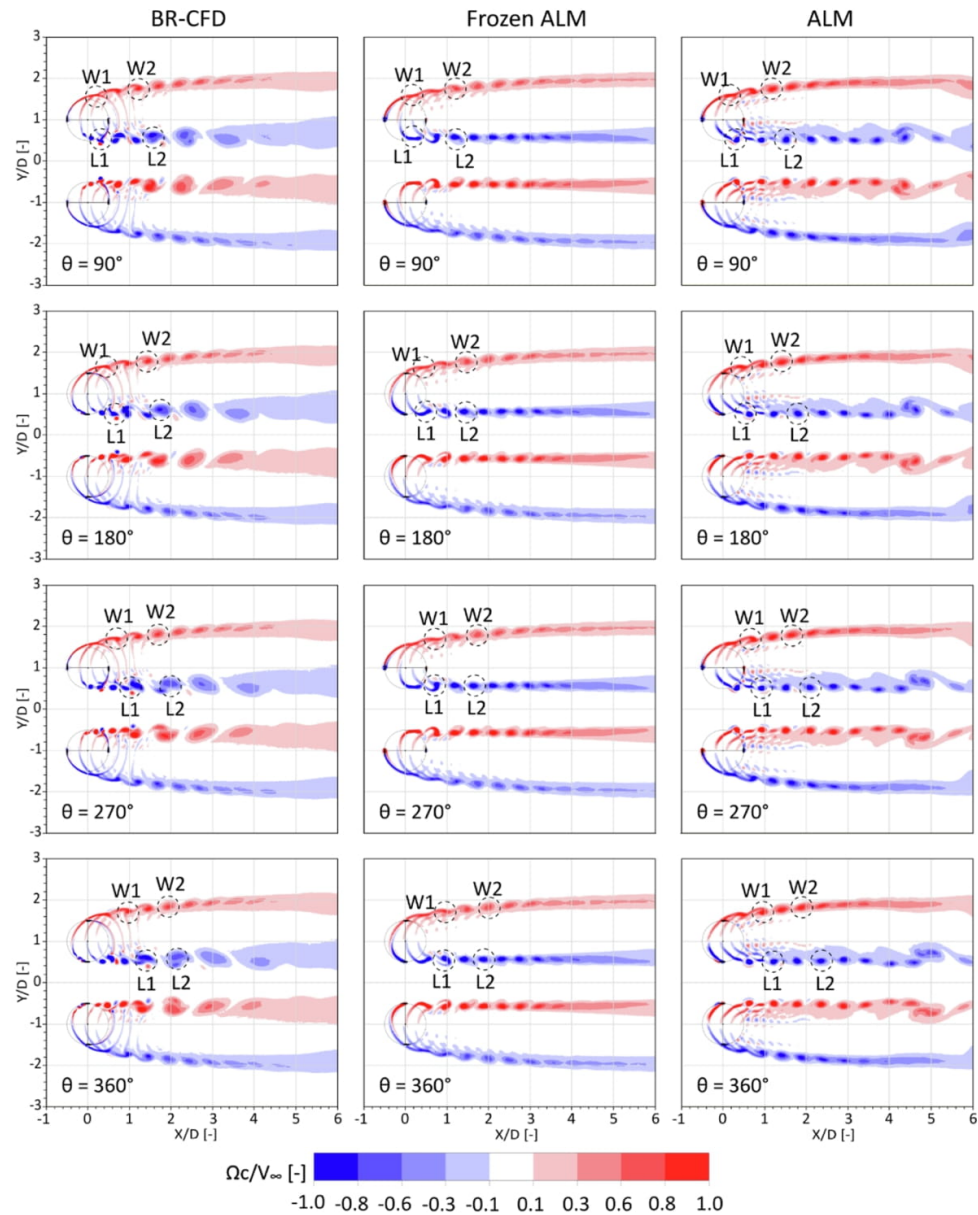


Fig. 13. Comparison between blade-resolved CFD and ALM at $\theta = 90^\circ$, 180° , 270° , and 360° in terms of dimensionless vorticity for 2B-VAHT twin rotors at $\text{TSR} = 2.5$.

the AoA was sampled using the same *LineAverage* technique for both ALM and blade-resolved CFD, with the settings described in Section 3.

For both TSRs, the trend of AoA variation predicted by the ALM show good agreement with that of the blade-resolved CFD, particularly at $\text{TSR} = 3$, exception made for a slight mismatch around $\theta \approx 200^\circ$. More importantly, the ALM captures the increase in the AoA induced in the gap region ($90^\circ < \theta < 225^\circ$) by the rotors' mutual interaction. This provides a first confirmation of what already observed for the 2B-VAWT turbine (see Section 4.1.2), i.e., that the ALM in the present formulation is able to reproduce with sufficient accuracy the flow pattern generated by a twin rotor configuration.

The discrepancy in terms of blade loads highlighted in Section 4.2.1 must therefore be related to their computation inside the ALM algorithm, in particular the selected model for unsteady aerodynamics and dynamic stall. This is clearly visible in the C_L profiles reported in Fig. 10. For $\text{TSR} = 2.5$, there is a good matching in terms of lift slope and stall delay between the two approaches, up to the maximum angle of attack ($\text{AoA} \sim -17^\circ$). In the *pitch-down* phase nonetheless, where most of the power augmentation effect takes place in the twin rotor case, the Berg dynamic stall model tends to overestimate the amplitude of the lift hysteresis loop ($-17^\circ < \text{AoA} < -5^\circ$), underestimating in turn the corresponding torque production (see Fig. 8). The opposite occurs in the

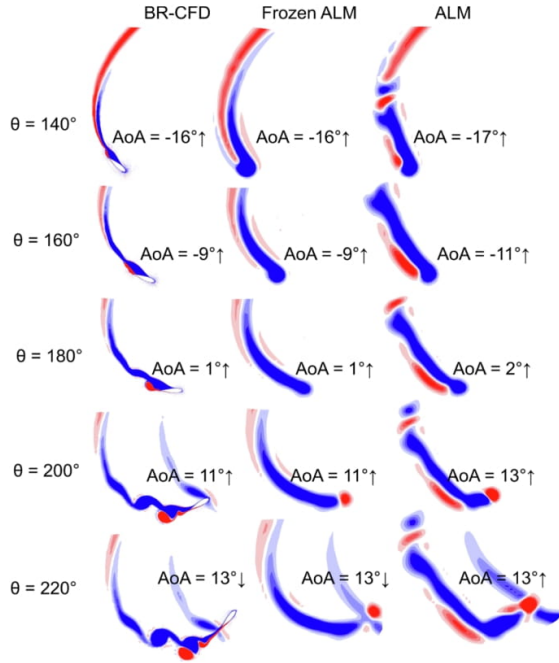


Fig. 14. Leeward vortex initiation for blade-resolved CFD compared with that of the Frozen ALM and ALM at TSR = 2.5.

downwind part, where the ALM keeps following a linear behaviour, while in the reference data there is both a reduction of the nominal slope and a lag of the instantaneous lift with respect to the angle of attack. This phenomenon is coherent with the airfoil behaviour observed during highly unsteady attached flow (as the one modelled in the theory of Theodorsen [49] for instance), and it is not included in the adopted Berg model. More advanced formulations are therefore required in the future. The weight of such unsteady loading, together with the discrepancy between ALM and blade-resolved CFD, is enhanced at TSR = 3, as separation-related effects lose importance.

4.2.3. Wake flow field

Fig. 12 presents a comparison of the non-dimensional velocity profiles, V/V_∞ , at various distances downstream of the rotors (1D, 3D, and 5D) obtained from ALM, Frozen ALM, and blade-resolved CFD simulations at TSRs 2.5 and 3. Overall, the velocity profiles of the ALM demonstrate good agreement with the reference profiles across all operating conditions. At TSR = 2.5, the ALM slightly overestimates the flow speed at the turbine sides ($Y/D \leq -2$ and $2 \leq Y/D$), while achieving a perfect match at TSR = 3. The presence of rotor-induced velocity deficits is accurately captured by the ALM for both TSRs, although the near wake velocity profile exhibits some variability at TSR = 2.5 and greater stability at TSR = 3. Within the mutual interaction region ($-0.5 \leq Y/D \leq 0.5$) at 1D and 3D, the ALM successfully captures the velocity variation, with minor differences compared to the blade-resolved CFD at TSR = 3. However, at 5D, the ALM solutions tend to overestimate the streamwise wake velocity between the rotors for both TSRs. Regarding the Frozen ALM, the velocity profiles closely match those of the blade-resolved CFD at the turbine leeward side ($Y/D < -1.5$ and $Y/D > 1.5$) at both TSRs. Although slight deviations are observed within the mutual interaction region, particularly near the rotors at $X/D = 1$ at TSR = 2.5.

Figs. 13 and 15 depict the dimensionless vorticity contours obtained from the ALM, Frozen ALM, and blade-resolved CFD solutions at TSRs 2.5 and 3. It is apparent that in all cases and at both TSRs, the twin rotors

produce symmetrical four vorticity sheets, each exhibiting varying directions due to the counter-rotating configuration. Each rotor generates two distinct shedding vortices which detach from the airfoil surface when subjected to a shift in the angle of attack sign. To analyze the evolution of these sheets, they were categorized as windward and leeward sheets. At each side, two vortices were identified and denoted as W1 and W2 for the windward vortices, and L1 and L2 for the leeward vortices. It is important to note that the identified vortices are generated by the same blade and correspond to two subsequent revolutions, while the ones in between have been generated by the other blade.

At TSR = 2.5, W1 initiates at $\theta = 90^\circ$ with a consistent slender structure across all cases. It gradually transforms into an elliptical shape until reaching $\theta = 360^\circ$, where the W1 vortices of the blade-resolved CFD, Frozen ALM, and ALM simulations coincide at $X/D \approx 1$. However, it is evident that the ALM vortex appears more compact compared to those of the Frozen ALM and blade-resolved CFD. This disparity becomes more pronounced from W2, being located at $X/D \approx 1.2$ at $\theta = 90^\circ$ and subsequently progressing to $X/D \approx 2$ at $\theta = 360^\circ$ in all simulations. Despite the similarity in shape and transport velocities between the windward vortices in all cases, their dissipation is different, especially at $X/D \approx 3$. In the case of ALM, the dissipation of the windward vortices assumes a slender form with a higher vorticity magnitude. Conversely, the windward vortices of the blade-resolved CFD exhibit a wider, diffused shape with a lower vorticity magnitude. Interestingly, the Frozen ALM initially displays windward vortices similar to those of the blade-resolved CFD until approximately $X/D \approx 4$, after which it demonstrates distinct dissipation behavior.

Regarding the leeward vortices at TSR = 2.5, it is apparent that their structures differ across all simulations from the initiation phase. This disparity can be attributed to two primary reasons. Firstly, there are discrepancies in the ALM load estimation at $\theta > 100^\circ$ (see Fig. 10), which is the azimuthal region where these vortices are formed. Secondly, the projection of forces, which the ALM is inherently unable to reproduce, particularly those formed around the blade wall in the blade-resolved CFD case. The latter is depicted in detail in Fig. 14. In the blade-resolved CFD results, due to the dynamic unsteadiness, a separation bubble is formed on the trailing edge, extending up to almost half of the airfoil surface. This results in the formation of four vortices in each half cycle. At $\theta = 90^\circ$, around L1, there are two additional blue vortices alongside a counter-rotating red vortex. These vortices are not transported at the same speed, and this disparity becomes more pronounced at $\theta = 180^\circ$ when L1 and its accompanying counter-rotating vortex interact with another vortex from the previous cycle, gradually dissipating the counter-rotating vortex. This phenomenon is more clearly visible with L2, where at $\theta = 90^\circ$ it pairs with another co-rotating vortex, leading to the complete disappearance of the counter-rotating vortex at $\theta = 270^\circ$. Subsequently, these vortices interact to form a larger single vortex, which eventually dissipates, forming a thick layer of flow with a low vorticity magnitude.

In the case of the Frozen ALM, despite projecting the force magnitudes from the blade-resolved CFD case, the resulting vortical structures differ due to the different distribution of these forces when projected. In the Frozen ALM, the forces take on a circular shape. On the other hand, the ALM generates a leeward street with more concentrated circular-shaped vortices. These vortices maintain their shape until $X/D \approx 5$, where they slow down and pair with each other, leading to their dissipation along with the windward sheet. It is important to note that in both ALM and blade-resolved CFD simulations, the leeward vortices pair together, forming larger vortices. However, the location of these pairings differs between the simulations: $1 < X/D < 2$ for the blade-resolved CFD and $4 < X/D < 5$ for the ALM. In contrast, no vortex pairing occurs in the Frozen ALM case.

At TSR = 3, the windward vortices look similar to those of TSR = 2.5 for all cases, with a single vortex produced every half cycle and a more compact structure for the ALM. The dissipation behavior is similar for all cases, with the ALM and Frozen ALM forming a thin layer of high

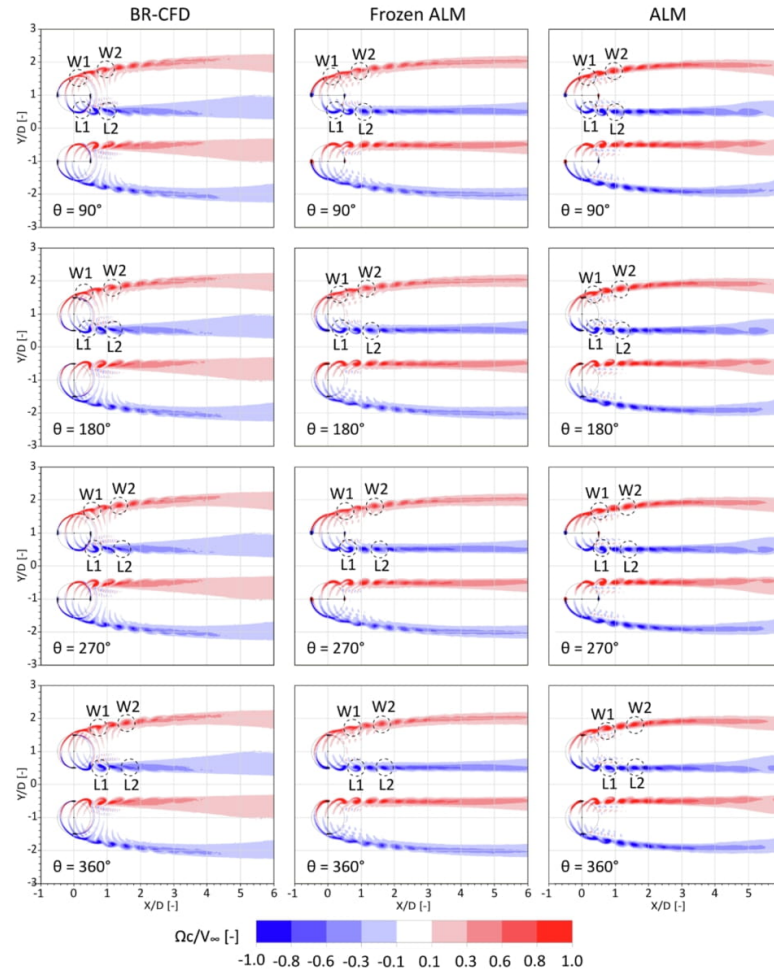


Fig. 15. Comparison between blade-resolved CFD and ALM at $\theta = 90^\circ, 180^\circ, 270^\circ$, and 360° in terms of dimensionless vorticity for 2B-VAHT twin rotors at $TSR = 3$.

vorticity flow that only in the ALM case, they break down into larger vortices upon interacting with the leeward street, more apparent at $X/D > 7$. In contrast, the blade-resolved CFD produces a thicker and more diffused windward street. Regarding the leeward vortices, due to the low angle of attack variation range at this TSR (see Fig. 9), the vortical structures formed on that side seem simpler compared to those at $TSR = 2.5$, with only a single vortex formed each half cycle, which quickly dissipates as shown from $L2$ at $\theta = 360^\circ$. However, as noted before, due to the poor load estimation at $\theta > 70^\circ$ (see Fig. 8), the leeward streets produced by the ALM have a completely different shape and concentration compared to those of the blade-resolved CFD and *Frozen ALM*, leading to different recovery behavior, which is more prominent at $X/D \approx 6$.

To summarize, the proposed tool is able to reproduce the main features of the wake flow field in a fairly accurate manner. However, the ALM shows a slower diffusion in the windward vortex streets at both $TSRs$. This can be mainly attributed to the discrepancies in the load estimation that the ALM produces during the blade pitch down phase. It is important to note that this has also been observed in [22] for a stand-alone rotor. Based on the analysis above, the POD region was chosen to be within a streamwise distance of $1 \leq X/D \leq 5$, since beyond this range, it is already evident that the flow behaves completely differently in both

simulations.

Fig. 16 presents a comparison of the percentage of the POD modal energy captured by each spatial mode, as well as the cumulative total energy captured up to each mode, at two different $TSRs$ of 2.5 and 3, calculated as explained in Section 3.4. At $TSR = 2.5$, the ALM simulation required 8 modes to capture over 90% of the total energy, while only 2 and 5 modes were needed in the case of *Frozen ALM* and blade-resolved CFD, respectively. At $TSR = 3$, due to the reduced influence of unsteady effects, more energy was captured by fewer modes in all simulations. Notably, the *Frozen ALM* and blade-resolved CFD exhibited a close match, with the first two modes capturing 96% and 94% of the total modal energy, respectively. In contrast, for the ALM, the first two modes accounted for only 67% of the total energy. These differences in the distribution of flow modal energy among the various modes between the ALM and blade-resolved CFD can be better understood by examining the corresponding modal spatial patterns.

Fig. 17 illustrates the normalized mean velocity fields at $TSR = 2.5$ obtained through the three simulation techniques, along with the contour representing the mean velocity differences calculated by subtracting the mean velocity of the ALM fields from that of the blade-resolved CFD. Overall, the mean flow patterns reveal a considerable similarity in the distribution of the wake flow field, with only some noticeable

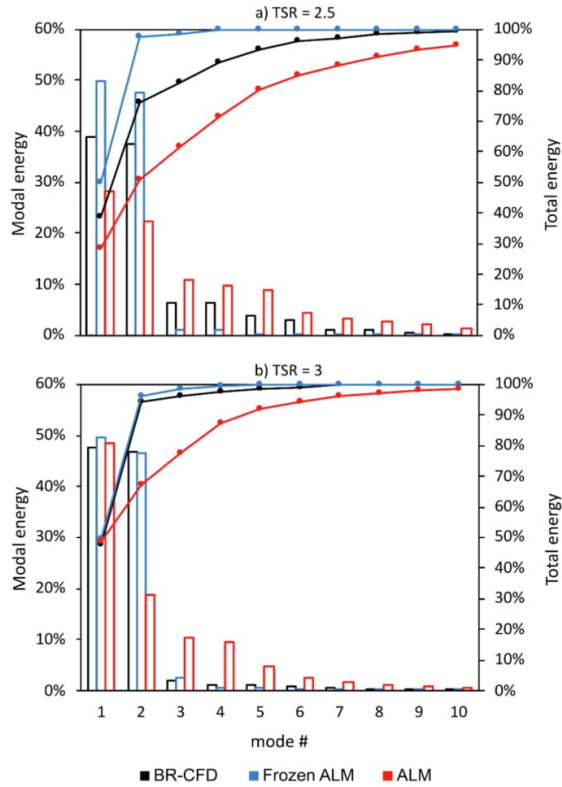


Fig. 16. Comparison between the percentage of modal energy for the ALM and blade-resolved CFD at TSRs a) 2.5 and b) 3.

differences highlighted by the mean velocity difference contour. A comparison between the Frozen ALM mean flow and the blade-resolved CFD demonstrates a close alignment in most regions, except for the wake borders where deviations occur. In contrast, the ALM mean flow exhibits greater acceleration in the area between the two rotors, as also observed in the velocity profiles presented in Fig. 12.

The first four spatial modes resulted from the POD are shown in Fig. 18, offering further insights into the differences among the three simulations in terms of wake development at $TSR = 2.5$. Notably, the first four modes in all simulations exhibit a pattern where each pair of modes shows a spatial phase shift between them. This phenomenon has also been observed in a recent single rotor POD analysis conducted in [47]. In the case of blade-resolved CFD, the first two modes represent the dominant modes of velocity fluctuations resulting from the coupling of the windward streets, while the other pair captures the harmonics of these windward streets, featuring double the number of structures, along with structures associated with the leeward vortex streets. Conversely, as previously explained the Frozen ALM results do not exhibit this coupling between the windward vortices, despite projecting the same forces as in the blade-resolved CFD (see Fig. 13). This explains the absence of modes 1 and 2 from the Frozen ALM modes, while modes 1 and 2 of the Frozen ALM align with modes 3 and 4 of the blade-resolved CFD.

Additionally, the first two modes of the ALM capture the fluctuations resulting from the pairing of the windward streets, where large structures are observed within $4 < X/D < 5$, indicating the formation of a double-sized vortex due to an earlier pairing. It is important to note that the first two modes of both simulations are related to the convection of the windward streets. However, this represents a significant difference between the wake flow fields of the ALM and the blade-resolved CFD, as discussed earlier based on the vorticity flow field. On the other hand, the third and fourth modes of the ALM exhibit paired velocity fluctuation patterns arising from the shedding vortices near both the leeward and windward sides. From $X/D = 1$ up to $X/D \approx 3$, modes 3 and 4 of the ALM exhibit strong similarities with modes 3 and 4 of the blade-resolved CFD, particularly in terms of the leeward streets.

Fig. 19 displays the mean velocity contours for blade-resolved CFD, Frozen ALM, and ALM, as well as the mean velocity differences at $TSR =$

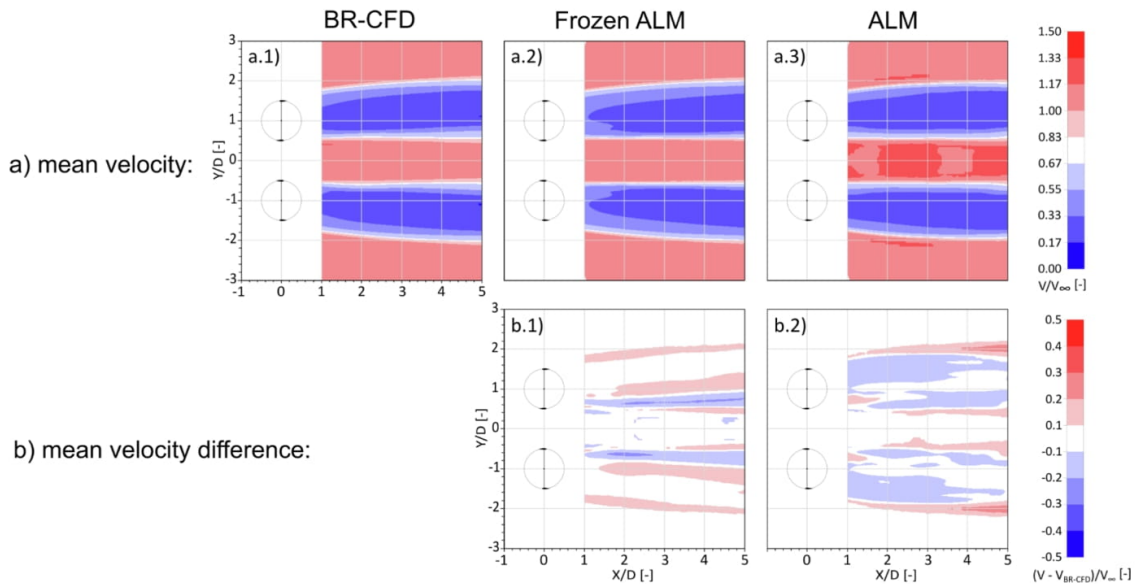


Fig. 17. Comparison between a) temporal mean of dimensionless velocity, b) mean velocity difference for 2B-VAHT twin rotors at $TSR = 2.5$.

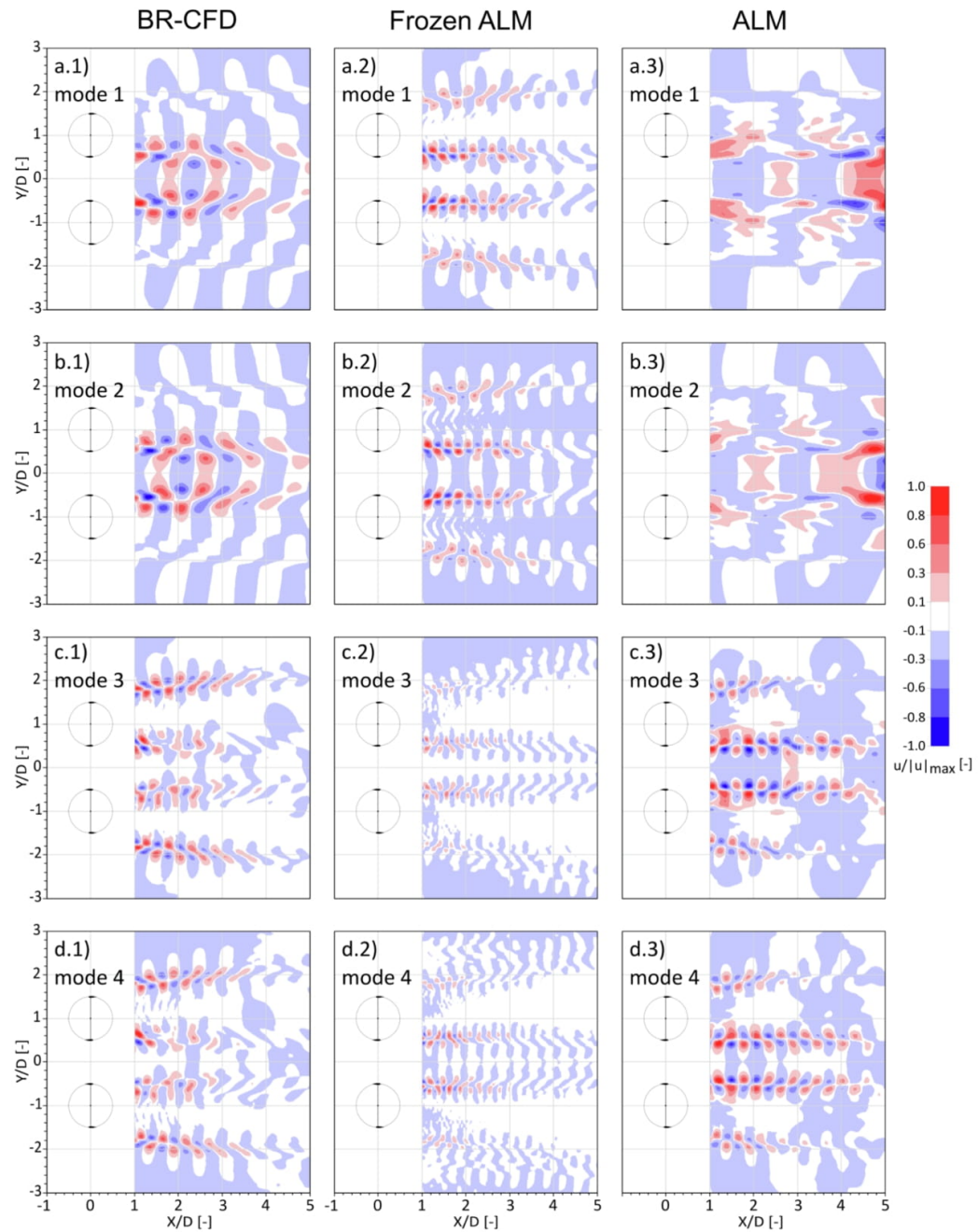


Fig. 18. Comparison between POD modes: a) mode 1, b) mode 2, c) mode 3, and d) mode 4 for 2B-VAHT twin rotors at $TSR = 2.5$.

3. At this TSR , the mean flow in the ALM simulation shows greater acceleration than in the blade-resolved CFD on both the leeward and windward sides, which was previously observed in the velocity profiles shown in Fig. 12. The Frozen ALM, on the other hand, exhibits higher acceleration within the area of mutual interaction between the wakes of the two turbines at $2 < X/D < 4$.

Fig. 20 shows the initial four POD modes, the first two modes of both

the blade-resolved CFD and the Frozen ALM show pairs of similar coherent structures with a streamwise phase shift, representing the velocity fluctuation pattern due to the shedding streets in both leeward and windward directions. The size and position of the coherent structures in the first two modes match well between the blade-resolved CFD and the Frozen ALM. For these two simulations, the 3rd mode shows a coherent pattern formed due to the dissipation of the leeward streets, while the

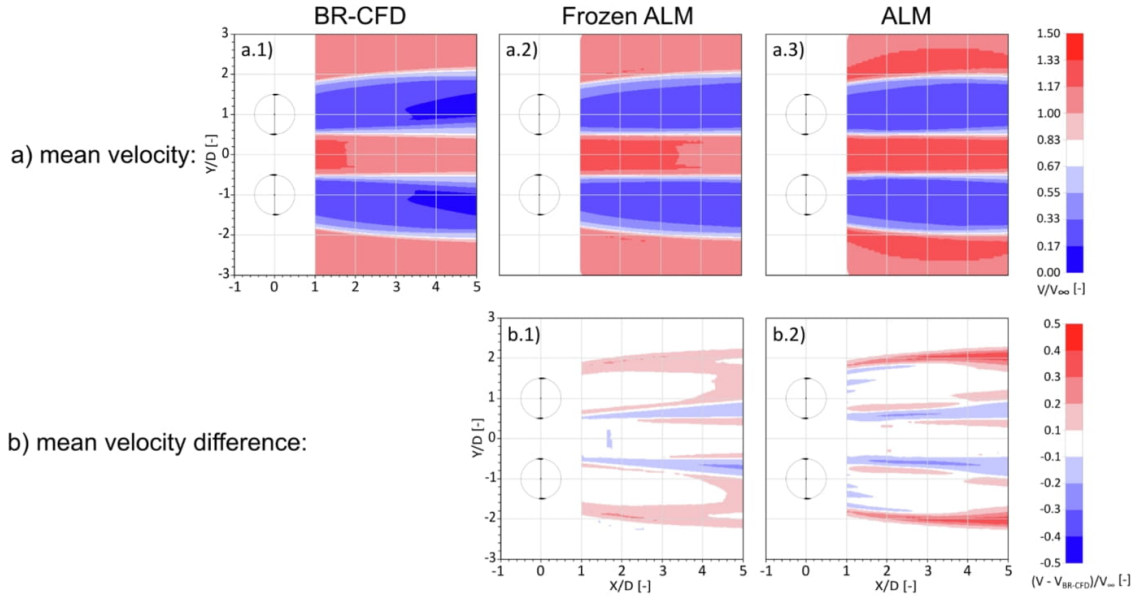


Fig. 19. Comparison between a) temporal mean of dimensionless velocity, b) mean velocity difference for 2B-VAHT twin rotors at TSR = 3.

4th mode represents a breakdown of the first and second modes, showing double the number of coherent structures. At this TSR, the influence of the 3rd and 4th modes is marginal since the first two modes capture about 94% and 96% of the modal energy for the blade-resolved CFD and the Frozen ALM, respectively (see Fig. 16).

For the ALM, the first two modes represent a pair of coherent structures that are produced near the rotors due to the windward streets. The 3rd and 4th modes depict a breakdown of these windward structures in addition to those formed due to the leeward streets. Interestingly, the 3rd and 4th modes of the ALM show good agreement with the 1st and 2nd modes of the blade-resolved CFD and the Frozen ALM.

5. Discussion

In this study, the ALM simulation approach was thoroughly validated against blade-resolved CFD solutions to evaluate its potential in accurately predicting the performance of twin Darrieus turbines. The validation process focused on three key aspects: (i) estimation of the instantaneous turbine loads, (ii) ability to mimic the local flow kinematics, and (iii) reproduction of the wake behaviour.

The results from the validation campaign have highlighted that estimating the instantaneous blade load is a task that is highly dependent on the specific case, as previously mentioned in [22]. It is evident that this issue is not exclusive to the implementation of multiple rotors but is influenced by the geometric and operational characteristics of the turbines. Specifically, the analysis of the instantaneous torque coefficient results for the 3B-VAWT turbines indicated that the ALM consistently overestimated the torque coefficient during the downwind azimuthal path, although this discrepancy was less pronounced at higher tip speed ratios. Conversely, for the 2B-VAHT pair, this issue persisted even at higher tip speed ratios. The primary reason for this disparity can be attributed to the incapability of the Berg model to accurately capture the unsteady effects in the absence of dynamic stall. These unsteady effects may not have as significant an impact on the 3B-VAWT rotor due to differences in operational characteristics, such as the lower c/R ratio and the higher solidity.

In terms of the local variation of the flow field, which has been previously identified [29] as the primary source of efficiency

improvement when two rotors are closely deployed, our findings reveal that the ALM successfully reproduces this effect. Although the data for the angle of attack was not available for the 3B-VAWT pair, we compared the velocity profiles in the vicinity of the blades with the results published by [5]. For the 2B-VAHT rotor pair, the AoA values generated by the ALM closely matched those obtained from the blade-resolved CFD simulations, particularly at higher TSR. These findings readily suggest that the ALM has the potential to be employed in multirotor configurations, as it can capture the corresponding array effects.

The comparative analysis using POD on the 2B-VAHT twin rotors revealed that the ALM simulation could effectively capture the main flow patterns shown by blade-resolved CFD, despite using different temporal and spatial discretisation's. The POD analysis played a crucial role in identifying the dominant flow patterns associated with the wake behind the twin rotors and highlighting the differences between the wakes produced by both simulations. In this context, the use of the *Frozen ALM* in the POD analysis allowed the distinction between the discrepancies due to the load estimation and those due to other factors such as the projection function. For instance, at high TSR, where the dynamic effects are minimal, the dominant modes of both blade-resolved CFD and Frozen ALM matches in terms of energy content as well as the coherent structures. Furthermore, the analysis clearly demonstrated that the discrepancies observed in the ALM reproduced wake were primarily due to the underestimation of the load generated during blade pitch down, which implicitly affected the wake recovery behavior.

Overall, the ALM has demonstrated exceptional capability in reproducing changes in local flow kinematics, which is essential for closely-spaced Darrieus turbine applications [29]. This has made it a suitable tool for interdisciplinary analyses of multiple Darrieus rotor problems, particularly for design and optimization purposes.

The spatial domain of the ALM is smaller than that of the blade-resolved CFD simulation ($42D \times 90D$ vs. $30D \times 45D$), with approximately 180 k elements compared to 1,000 k elements. Additionally, the ALM uses a larger $\Delta\theta$ (1°) than the blade-resolved CFD (0.4°), resulting in a significant reduction in computational time. The ALM takes approximately 7 core-hours per revolution, while the blade-resolved CFD takes around 508 core-hours per revolution.

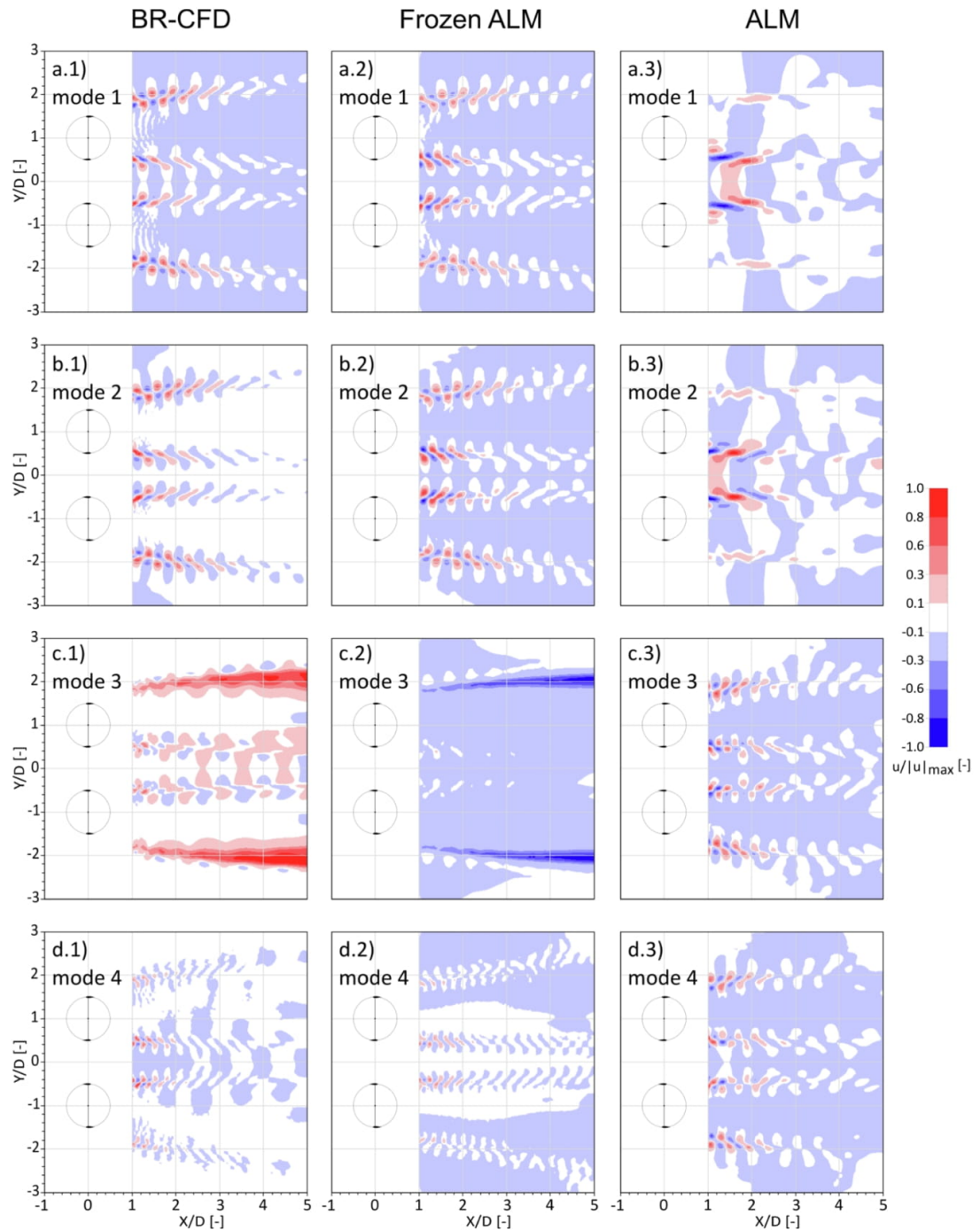


Fig. 20. Comparison between POD modes: a) mode 1, b) mode 2, c) mode 3, and d) mode 4 for 2B-VAHT twin rotors at $TSR = 3$.

6. Conclusions

This study has presented a comprehensive validation campaign of the ALM tool in its two-dimensional form for closely-spaced twin

Darrieus turbines. Proving that this computationally-efficient method is able to capture the complex flow interactions, which have been to date analyzed only by means of blade-resolved CFD, is thought to represent a finding of particular relevance for industry, as this method can become

the new standard in the growing sector, for example, of hydrokinetic turbines that need to be deployed into arrays.

The validation was conducted by comparing the results of the ALM with those of blade-resolved CFD simulations, and it aimed at assessing the ALM's ability to estimate the instantaneous turbine loads, mimic the local flow kinematics, and reproduce the wake behaviour. The validation campaign was carried out for two different pairs of Darrieus turbines: *3B-VAWT* and *2B-VAHT*, which exhibit different geometrical and operational characteristics.

The results of the validation campaign have demonstrated that the ALM is able of accurately reproducing the local flow kinematics, which is essential for closely-spaced Darrieus turbine applications. The load analysis showed that the ALM can capture the instantaneous load with good to fair accuracy, although this may be case-dependent due to the sensitivity of the adopted dynamic stall model to geometrical and operating conditions. The wake analysis conducted through performing POD analyses revealed that the main flow fluctuating patterns from the blade-resolved simulation are well captured by the ALM. In summary, the ALM requires much less computational resources compared to blade-resolved CFD for simulating a twin-rotor configuration, with a computational time difference of almost two orders of magnitude. This makes the ALM an attractive alternative for complicated and interdisciplinary applications, such as simulating the flow around a floating Darrieus wind turbine or examining the effect of Darrieus hydrokinetic turbines on a channel flow.

In future research, it would be beneficial to expand the validation of the ALM to encompass three-dimensional simulations with experimental validation. These simulations should consider factors such as tip losses and the impact of vertical mixing on wake recovery. Furthermore, incorporating additional physical phenomena into the ALM model would enhance our understanding of how these machines perform in realistic scenarios.

CRedit authorship contribution statement

Omar S. Mohamed: Methodology, Software, Validation, Formal analysis, Investigation, Data curation, Writing – original draft, Visualization. **Pier Francesco Melani:** Methodology, Software, Writing – review & editing, Visualization. **Francesco Papi:** Methodology, Software, Writing – review & editing. **Francesco Balduzzi:** Conceptualization, Methodology, Writing – review & editing. **Alessandro Bianchini:** Conceptualization, Methodology, Investigation, Resources, Data curation, Writing – review & editing, Supervision, Project administration, Funding acquisition.

Declaration of Competing Interest

The authors declare that they have no known competing financial interests or personal relationships that could have appeared to influence the work reported in this paper.

Data availability

No experimental data was used for the research described in the article. CFD results can be made available on request.

References

- [1] Tjiu W, Marnoto T, Mat S, Ruslan MH, Sopian K. Darrieus vertical axis wind turbine for power generation II: Challenges in HAWT and the opportunity of multi-megawatt Darrieus VAWT development. *Renew Energy* Mar. 2015;75:560–71. <https://doi.org/10.1016/j.renene.2014.10.039>.
- [2] Rolin V-F-C, Porté-Agel F. Experimental investigation of vertical-axis wind-turbine wakes in boundary layer flow. *Renew Energy* Apr. 2018;118:1–13. <https://doi.org/10.1016/j.renene.2017.10.105>.
- [3] Balduzzi F, Melani PF, Soraperra G, Brighenti A, Battisti L, Bianchini A. Some design guidelines to adapt a Darrieus vertical axis turbine for use in hydrokinetic applications. *E3S Web Conf* 2021;312:08017. <https://doi.org/10.1051/e3sconf/202131208017>.
- [4] Dabiri JO. Potential order-of-magnitude enhancement of wind farm power density via counter-rotating vertical-axis wind turbine arrays. *J Renewable Sustainable Energy* 2011;3(4):043104. <https://doi.org/10.1063/1.3608170>.
- [5] Zanforlin S, Nishino T. Fluid dynamic mechanisms of enhanced power generation by closely spaced vertical axis wind turbines. *Renew Energy* 2016;99:1213–26. <https://doi.org/10.1016/j.renene.2016.08.015>.
- [6] Mohamed OS, Ibrahim A, El Baz AMR. 'CFD investigation of the multiple rotors darrieus type turbine performance', presented at the ASME turbo expo turbomachinery technical conference and exposition. American Society of Mechanical Engineers Digital Collection 2019;2019. <https://doi.org/10.1115/GT2019-91491>.
- [7] Alexander AS, Santhanakrishnan A. Mechanisms of power augmentation in two side-by-side vertical axis wind turbines. *Renew Energy* 2020;148:600–10. <https://doi.org/10.1016/j.renene.2019.10.149>.
- [8] Zheng H-D, Zheng XY, Zhao SX. Arrangement of clustered straight-bladed wind turbines. *Energy* 2020;200:117563. <https://doi.org/10.1016/j.energy.2020.117563>.
- [9] Guilbot M, Barre S, Balarac G, Bonamy C, Guillaud N. 'A numerical study of vertical axis wind turbine performances in twin-rotor configurations'. *J Phys: Conf Ser* 2020;1618(5):052012. <https://doi.org/10.1088/1742-6596/1618/5/052012>.
- [10] Sahebzadeh S, Rezaeiha A, Montazeri H. Vertical-axis wind-turbine farm design: Impact of rotor setting and relative arrangement on aerodynamic performance of double rotor arrays. *Energy Rep* Nov. 2022;8:5793–819. <https://doi.org/10.1016/j.egy.2022.04.030>.
- [11] 'Comparison between two-dimensional and three-dimensional computational fluid dynamics techniques for two straight-bladed vertical-axis wind turbines in inline arrangement - Saeed Nazari, Mahdi Zamani, Sajad A Moshizi, 2018'. <https://journals.sagepub.com/doi/10.1177/0309524X18780384> (accessed Jun. 30, 2022).
- [12] Draper S, Nishino T. Centred and staggered arrangements of tidal turbines. *J Fluid Mech* 2014;739:72–93. <https://doi.org/10.1017/jfm.2013.593>.
- [13] Nishino T, Willden RHJ. The efficiency of an array of tidal turbines partially blocking a wide channel. *J Fluid Mech* Oct. 2012;708:596–606. <https://doi.org/10.1017/jfm.2012.349>.
- [14] Tavernier DD, Ferreira C, Li A, Paulsen US, Madsen HA. Towards the understanding of vertical-axis wind turbines in double-rotor configuration. *J Phys: Conf Ser* 2018;1037. <https://doi.org/10.1088/1742-6596/1037/2/022015>.
- [15] Gauvin-Tremblay O, Dumas G. Hydrokinetic turbine array analysis and optimization integrating blockage effects and turbine-wake interactions. *Renew Energy* 2022;181:851–69. <https://doi.org/10.1016/j.renene.2021.09.003>.
- [16] Sorensen JN, Shen WZ. Numerical modeling of wind turbine wakes. *J Fluids Eng* 2002;124(2):393–9. <https://doi.org/10.1115/1.1471361>.
- [17] Shamsoddin S, Porté-Agel F. Large Eddy Simulation of Vertical Axis Wind Turbine Wakes. *Energies* 2014;7(2). <https://doi.org/10.3390/en7020890>.
- [18] P. Bachant, A. Goude, and M. Wosnik, 'Actuator line modeling of vertical-axis turbines', *arXiv:1605.01449 [physics]*, Sep. 2018, Accessed: Jun. 16, 2020. [Online]. Available: <http://arxiv.org/abs/1605.01449>.
- [19] Mendoza V, Bachant P, Ferreira C, Goude A. 'Near-wake flow simulation of a vertical axis turbine using an actuator line model'. *Wind Energy* 2019;22(2): 171–88. <https://doi.org/10.1002/we.2277>.
- [20] Melani PF, Balduzzi F, Ferrara G, Bianchini A. Tailoring the actuator line theory to the simulation of Vertical-Axis Wind Turbines. *Energy Convers Manage* 2021;243: 114422. <https://doi.org/10.1016/j.enconman.2021.114422>.
- [21] Melani PF, Balduzzi F, Ferrara G, Bianchini A. Development of a desmodynamic variable pitch system for hydrokinetic turbines. *Energy Convers Manage* 2021;250: 114890. <https://doi.org/10.1016/j.enconman.2021.114890>.
- [22] Mohamed OS, Melani PF, Balduzzi F, Ferrara G, Bianchini A. An insight on the key factors influencing the accuracy of the actuator line method for use in vertical-axis turbines: Limitations and open challenges. *Energy Convers Manage* 2022;270: 116249. <https://doi.org/10.1016/j.enconman.2022.116249>.
- [23] Hezaveh SH, Bou-Zeid E, Dabiri J, Kinzel M, Cortina G, Martinelli L. Increasing the power production of vertical-axis wind-turbine farms using synergistic clustering. *Boundary-Layer Meteorol* 2018;169(2):275–96. <https://doi.org/10.1007/s10546-018-0368-0>.
- [24] S. R. V. R. S. Solanki, V. K. Chalamalla, and S. S. Sinha, 'Numerical simulations and analysis of flow past vertical-axis wind turbines employing the actuator line method', presented at the Proceedings of the 26th National and 4th International ISHMT-ASTFE Heat and Mass Transfer Conference December 17–20, 2021, IIT Madras, Chennai-600036, Tamil Nadu, India, Begel House Inc., 2021. doi: 10.1615/IHMTC-2021.41110.
- [25] Zhang JH, Lien F-S, Yee E. 'Investigations of vertical-axis wind-turbine group synergy using an actuator line model'. *Energies* 2022;15(17). <https://doi.org/10.3390/en15176211>.
- [26] Melani PF, Balduzzi F, Bianchini A. A robust procedure to implement dynamic stall models into actuator line methods for the simulation of vertical-axis wind turbines. *J Eng Gas Turbines Power* 2021;143(11). <https://doi.org/10.1115/1.4051909>.
- [27] Melani PF, Balduzzi F, Ferrara G, Bianchini A. How to extract the angle attack on airfoils in cycloidal motion from a flow field solved with computational fluid dynamics? Development and verification of a robust computational procedure. *Energy Convers Manage* Nov. 2020;223:113284. <https://doi.org/10.1016/j.enconman.2020.113284>.
- [28] Mohamed OS, Elbaz AMR, Bianchini A. A better insight on physics involved in the self-starting of a straight-blade Darrieus wind turbine by means of two-dimensional

- Computational Fluid Dynamics. *J Wind Eng Ind Aerodyn* Nov. 2021;218:104793. <https://doi.org/10.1016/j.jweia.2021.104793>.
- [29] Mohamed OS, Melani PF, Balduzzi F, Ferrara G, Bianchini A. An insight on the physical mechanisms responsible of power augmentation in a pair of counter-rotating Darrieus turbines. *Energ Conver Manage* May 2023;284:116991. <https://doi.org/10.1016/j.enconman.2023.116991>.
- [30] Jost E, Klein L, Leipprand H, Lutz T, Krämer E. Extracting the angle of attack on rotor blades from CFD simulations. *Wind Energy* 2018;21(10):807–22. <https://doi.org/10.1002/we.2196>.
- [31] Azadani LN. Vertical axis wind turbines in cluster configurations. *Ocean Eng* 2023; 272:113855. <https://doi.org/10.1016/j.oceaneng.2023.113855>.
- [32] Cacciali L, Battisti L, Dell'Anna S. 'Multi-array design for hydrokinetic turbines in hydropower canals'. *Energies* 2023;16(5). <https://doi.org/10.3390/en16052279>.
- [33] Ferreira CS, Madsen HA, Barone M, Roscher B, Deglaire P, Arduin I. 'Comparison of aerodynamic models for Vertical Axis Wind Turbines'. *J Phys: Conf Ser* 2014;524: 012125. <https://doi.org/10.1088/1742-6596/524/1/012125>.
- [34] C. de Boor, *A Practical Guide to Splines*, in *Applied Mathematical Sciences*. New York: Springer-Verlag, 1978. Accessed: Nov. 23, 2020. [Online]. Available: <https://www.springer.com/gp/book/9780387953663>.
- [35] Migliore PG, Wolfe WP, Fanucci JB. Flow curvature effects on darrieus turbine blade aerodynamics. *Journal of energy* 1980;4(2):49–55. <https://doi.org/10.2514/3.62459>.
- [36] Rainbird JM, et al. On the influence of virtual camber effect on airfoil polars for use in simulations of Darrieus wind turbines. *EnergyConversion and Management* Dec. 2015;106:373–84. <https://doi.org/10.1016/j.enconman.2015.09.053>.
- [37] Drela M. 'XFOIL: An analysis and design system for low reynolds number airfoils'. In: *Conference on Low Reynolds Number Airfoil Aerodynamics*, University of Notre Dame; 1989. https://doi.org/10.1007/978-3-642-84010-4_1.
- [38] Viterna LA, Janetzke DC. Theoretical and experimental power from large horizontal-axis wind turbines. *NASA Tech Memo* 1982.
- [39] Marten D, et al. Effects of airfoil's polar data in the stall region on the estimation of Darrieus wind turbines performance. *J Eng Gas Turbines Power* 2016;139(2): 022606–9. <https://doi.org/10.1115/1.4034326>.
- [40] Massé B. Description de deux programmes d'ordinateur pour le calcul des performances et des charges aérodynamiques pour les éoliennes à axe vertical. technical report IREQ-2379 1981.
- [41] Martínez-Tossas LA, Churchfield MJ, Meneveau C. Optimal smoothing length scale for actuator line models of wind turbine blades based on Gaussian body force distribution. *Wind Energy* 2017;20(6):1083–96. <https://doi.org/10.1002/we.2081>.
- [42] F. R. Menter, 'Zonal Two Equation Kappa-Omega Turbulence Models for Aerodynamic Flows', presented at the AIAA Fluid Dynamics Conference, Orlando, FL, Jan. 1993. Accessed: Apr. 19, 2023. [Online]. Available: <https://ntrs.nasa.gov/citations/19960044572>.
- [43] Balduzzi F, Bianchini A, Maleci R, Ferrara G, Ferrari L. Critical issues in the CFD simulation of Darrieus wind turbines. *Renew Energy* 2016;85:419–35. <https://doi.org/10.1016/j.renene.2015.06.048>.
- [44] Balduzzi F, Bianchini A, Ferrara G, Ferrari L. Dimensionless numbers for the assessment of mesh and timestep requirements in CFD simulations of Darrieus wind turbines. *Energy* 2016;97:246–61. <https://doi.org/10.1016/j.energy.2015.12.111>.
- [45] Melani PF, Schito P, Persico G. 'Experimental assessment of an ACL simulation tool for VAWTs'. *Proc. of the TURbWind 2018 Colloquium*, Riva del Garda (Italy), September 6-7. 2018.
- [46] Taira K, et al. Modal analysis of fluid flows: An overview. *AIAA J* 2017;55(12): 4013–41. <https://doi.org/10.2514/1.J056060>.
- [47] Chen L, Xu J, Dai R. Numerical prediction of switching gurney flap effects on straight bladed VAWT power performance. *J Mech Sci Technol* Dec. 2020;34(12): 4933–40. <https://doi.org/10.1007/s12206-020-2106-z>.
- [48] Sheidani A, Salavatidezfouli S, Stabile G, Rozza G. Assessment of URANS and LES methods in predicting wake shed behind a vertical axis wind turbine. *J Wind Eng Ind Aerodyn* Jan. 2023;232:105285. <https://doi.org/10.1016/j.jweia.2022.105285>.
- [49] T. Theodorsen, 'General Theory of Aerodynamic Instability and the Mechanism of Flutter', NACA-TR-496, Jan. 1949. Accessed: Apr. 17, 2023. [Online]. Available: <https://ntrs.nasa.gov/citations/19930090935>.



Contents lists available at ScienceDirect

Ocean Engineering

journal homepage: www.elsevier.com/locate/oceaneng

Research paper

Three-dimensional CFD-ALM-VOF modeling of hydrokinetic turbines in realistic open-channel conditions

Omar S. Mohamed^a, Pier Francesco Melani^a, Giuseppe Soraperra^b, Alessandra Brighenti^b, Giovanni Ferrara^a, Vittorino Betti^b, Leonardo Schippa^c, Massimo Guerrero^d, Francesco Balduzzi^a, Alessandro Bianchini^{a,*}

^a Department of Industrial Engineering, Università degli Studi di Firenze, Via di Santa Marta 3, 50139, Firenze, Italy

^b HE-PowerGreen s.r.l., Via Solteri 74, 38121, Trento, Italy

^c Department of Civil Engineering, Università degli Studi di Ferrara, Via Saragat 1, 44122, Ferrara, Italy

^d Department of Civil, Chemical, Materials and Environmental Engineering, Università di Bologna, Viale del Risorgimento 2, 40136, Bologna, Italy

ARTICLE INFO

Keywords:

Actuator line model (ALM)
Darrieus
Vertical axis turbine (VAT)
Hydrokinetic turbines
Computational fluid dynamics (CFD)
Volume of fluid (VOF)

ABSTRACT

The potential of Darrieus-type Vertical Axis Turbines (VATs) in hydrokinetic applications, especially when deployed into arrays or clusters, has gained recognition in the field of distributed energy production. Due to the complex unsteady hydrodynamics of these turbines and the different flow scales involved, numerical studies to date have often focused on one system component only (turbine or channel). This study presents an integrate, accurate, but computationally feasible simulation approach applied to an array of VATs in a free-surface channel. The turbines are modelled using the Actuator Line Method (ALM), while the water free surface is modelled using the Volume of Fluid (VOF) approach, both included within the RANS solver of Ansys Fluent. A sensitivity analysis is conducted on the most appropriate model settings, followed by a validation campaign with field data that showed good agreement between experiments and simulations. Results point out the significance of the two-way interactions between the turbines and the channel, emphasizing the impact of the variation in water level and inflow velocity on the efficiency assessment of hydrokinetic turbines. The study also exploits the current 3D model to get a better understanding of the interaction effects between closely spaced turbines, and their influence on the power augmentation mechanisms in an array of turbines, which were so far mainly analyzed only with 2D approaches. Finally, the hydraulic impact on the channel is discussed, highlighting the effects and limitations of clustering hydrokinetic arrays in a confined channel.

1. Introduction

In conventional hydroelectric power generation, potential energy from water flow is harnessed as it falls from a considerable height. The energy produced in this process relies on two factors: the flow rate and the head of water. However, despite its superior efficiency over other renewable energy resources, this method is dependent on the presence of suitable morphological characteristics of the territory and often limited by economic, environmental, and ecological concerns (Yadav et al., 2023). Alternatively, interest is recently arising towards Hydrokinetic Turbines (HKTs), which are used to capture the kinetic energy from water streams in small channels and artificial water networks.

Rated power outputs are generally small, but these systems can work well in combination with other renewables in the perspective of a distributed generation, are easy to regulate and have negligible impact on the surrounding environment.

Similarly to wind turbines, HKTs can be categorized based on the orientation of the axis of rotation with respect to the incoming flow, i.e., Horizontal Axis Turbines (HATs) and Vertical Axis Turbines (VATs). Although less efficient than HATs (Niebuhr et al., 2019), VATs have the advantage of easier maintenance due to the possibility of placing the generator outside of the water. On top of that, deploying two or more VATs in close proximity can lead to increased overall power production compared to stand-alone turbines, both for Darrieus type (Zanforlin and

* Corresponding author.

E-mail addresses: omar.sherif@unifi.it (O.S. Mohamed), pierfrancesco.melani@unifi.it (P.F. Melani), gsoraperra@he-powergreen.it (G. Soraperra), amministrazione@he-powergreen.it (A. Brighenti), giovanni.ferrara@unifi.it (G. Ferrara), studiobettieviali@gmail.com (V. Betti), leonardo.schippa@unife.it (L. Schippa), massimo.guerrero@unibo.it (M. Guerrero), francesco.balduzzi@unifi.it (F. Balduzzi), alessandro.bianchini@unifi.it (A. Bianchini).

<https://doi.org/10.1016/j.oceaneng.2024.119411>

Received 19 July 2024; Received in revised form 11 September 2024; Accepted 30 September 2024

Available online 9 October 2024

0029-8018/© 2024 The Authors. Published by Elsevier Ltd. This is an open access article under the CC BY-NC-ND license (<http://creativecommons.org/licenses/by-nc-nd/4.0/>).

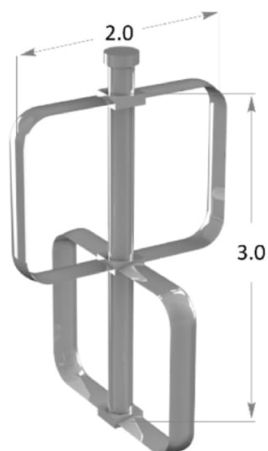


Fig. 1. a) schematic representation of the turbine model and b) blade profile.

Table 1
Geometrical characteristics of the test case turbine.

Dimension	Value
Number of blades [-]	2
Diameter, D [m]	2
Turbine height, H [m]	3

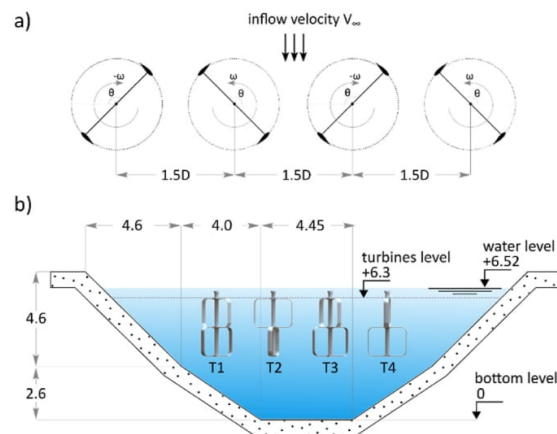


Fig. 2. Biffis channel cross section and dimensions in meters.

Table 2
Test operating conditions.

Parameter	Value
Tip-speed ratio [-]	2.09–3.15
Work fluid [-]	water
Inflow speed, V_∞ [m/s]	1.6
Inflow turbulence intensity [%]	0.24
Chord-based Reynolds number range, Re_c [-]	$\approx 420k - 1,600k$

Nishino, 2016; Mohamed et al., 2019, 2023a) and Savonius type (Nag and Sarkar, 2021; Chen et al., 2022). (Zanforlin and Nishino, 2016), (Mohamed et al., 2019), (Mohamed et al., 2023a). (Nag and Sarkar, 2021)., (Chen et al., 2022)

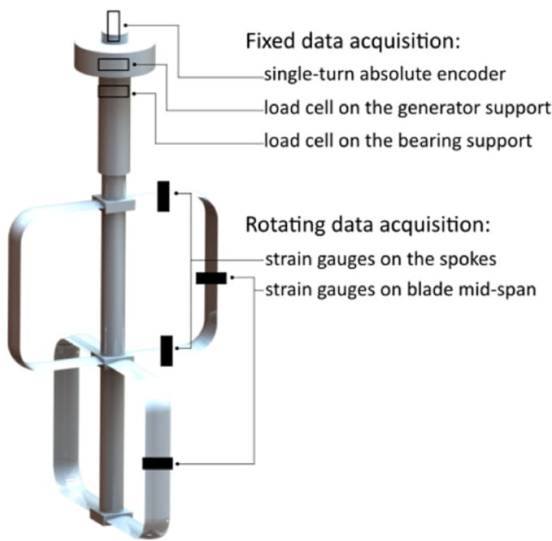


Fig. 3. Locations of the sensors used during the performance measurements experimental campaign.

The achievable power augmentation thanks to array effects, nonetheless, is limited by the interaction between HKTs and the channel hydraulic system. Several studies have investigated this phenomenon in subcritical flows conditions. For instance, an experimental campaign has been carried out by Mayers and Bahaj (Myers and Bahaj, 2007) on a HKT operating in a recirculating channel. They observed a rise in water level relative to the free surface near the upstream due to blockage, a decrease near the downstream due to energy extraction, followed by a subsequent rebound in the far downstream due to wake recovery. These observations have been confirmed in other studies (e.g. (Kirke, 2019; Chaulagain et al., 2023)., (Kirke, 2019)., (Chaulagain et al., 2023)

The prediction of this turbine-channel interaction is, however, a challenging task, due to the intrinsically unsteady operation, massively involving stall and creation of vortical structures, that characterizes Darrieus turbines. Several analytical models based on the Blade Element Momentum (BEM) were recently introduced to describe the performance of an array of Darrieus HKTs, often relying on the Double Multiple Streamtube (DMST) approach. Nishino and Willden (2012) developed an analytical model based on the actuator disk theory, to which they thereafter added a sub-model to account for the effect of array-scale flow expansion (Nishino and Willden, 2013). Their model was able to describe general trends related to the relation between the number of turbines within the array and the limit of power extraction per turbine. Draper and Nishino (2014) further developed the model to study centered and staggered arrays. Cacciali et al., 2022, 2023a developed a DMST-based model that iteratively updates the inflow upstream to a Darrieus HKT based on the downstream conditions in a free surface channel environment. In general, these analytical models are usually used as a rapid tool for the preliminary design, given that the BEM-based models in VATs are inherently incapable of predicting second-order phenomena. i.e., vortex shedding and blade-wake interaction, as well as three-dimensional effects. (Cacciali et al., 2022)., (Cacciali et al., 2023a)

On the other hand, Computational Fluid Dynamics (CFD) simulations come as a reliable tool for predicting such complex phenomena, however, with far more computational effort compared with analytical models. During the last couple of decades, CFD has played an important role in understanding the aero/hydrodynamic characteristics of these machines (Mohamed et al., 2021, 2023a; Melani et al., 2020). It is

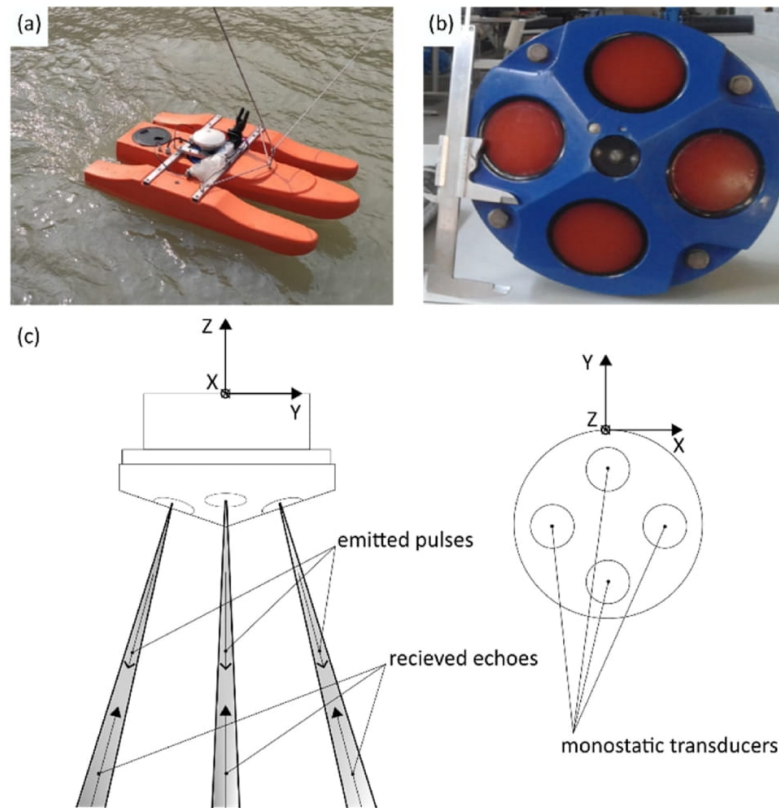


Fig. 4. (a) The boat used to mount the ADCP, b) ADCP instrument, c) schematic representation of the ADCP working concept.

important to note that due to constraints related to computational cost, most of these studies employed a two-dimensional approach without considering the effect of the two-way interactions between the turbines and the channel. However, these interactions in a three-dimensional domain need to be taken in consideration in several important aspects connected to the overall behaviour of these machines. For example, Nishi et al. (2017) and, recently, Yosry et al. (2023) conducted blade-resolved CFD studies for Darrieus HKT using Volume of Fluid (VOF) to model the free water surface, and they both showed that a single-phase simulation of a hydrokinetic turbine results in significantly overestimated power output compared to a two-phase simulation. (Mohamed et al., 2023a), (Melani et al., 2020), (Mohamed et al., 2021)

Hybrid simulation techniques, such as the Actuator Line Method (ALM) were recently proposed to substitute the conventional blade-resolved CFD in the simulations of Darrieus turbines (Melani et al., 2021a). In ALM, blade loads are estimated based on the sampled local inflow conditions using tabulated polar data as in engineering methods. These forces are then inserted into the computational grid as equivalent volume momentum sources. A CFD-based method is then used to solve the flow field. Since the geometry of the turbine blade is not explicitly resolved, but instead modelled using a lumped-parameter representation, the overall grid density of the simulation is significantly reduced, resulting in a corresponding decrease in computational cost.

The VOF method, as detailed in (Yosry et al., 2023; Gauvin-Tremblay and Dumas, 2020), when coupled with blade-resolved CFD, has proven effective for simulating the free water surface in hydrokinetic turbines, addressing crucial aspects of free-surface modeling. Building on this foundation and the authors' previous research, particularly regarding

the development and refinement of an ALM tool, tailored to VATs (Melani et al., 2021a) both isolated (Mohamed et al., 2022) and in array configurations (Mohamed et al., 2023b), the present study introduces a novel framework that bridges the gap between BEM-based models and blade-resolved CFD. This framework is specifically designed for the simulation of hydrokinetic turbine arrays under realistic channel flow conditions. It couples the ALM tool with a two-phase, three-dimensional model of a realistic water channel that employs the VOF approach to accurately capture free-surface effects (Gauvin-Tremblay and Dumas, 2020).

A systematic sensitivity analysis to optimize the numerical model and understand its limitations is performed, followed by a validation campaign against unique experimental field measurements on an industrial 4-rotors array operating in a channel in Italy (Guerrero et al., 2023). This validation involves performance, water level, and flow field data and is considered to be a major point of strength of the study. The load results of the array are then analyzed, followed by an assessment of the corresponding power augmentation mechanism. The results are compared with 2D findings to gain a deeper understanding of how closely spaced deployment can enhance the power coefficient in a confined channel within a realistic three-dimensional context. Finally, the impact of the array deployment on the channel is investigated, highlighting the potential and drawbacks of such turbine installations.

The paper is organized as follows. In Section 2, the hydrokinetic turbine model featured in this study is presented along with the configurations studied for the array configuration in addition to the geometrical and hydraulic characteristics of the channel. Section 3 details the methodology employed in the present study, including the ALM

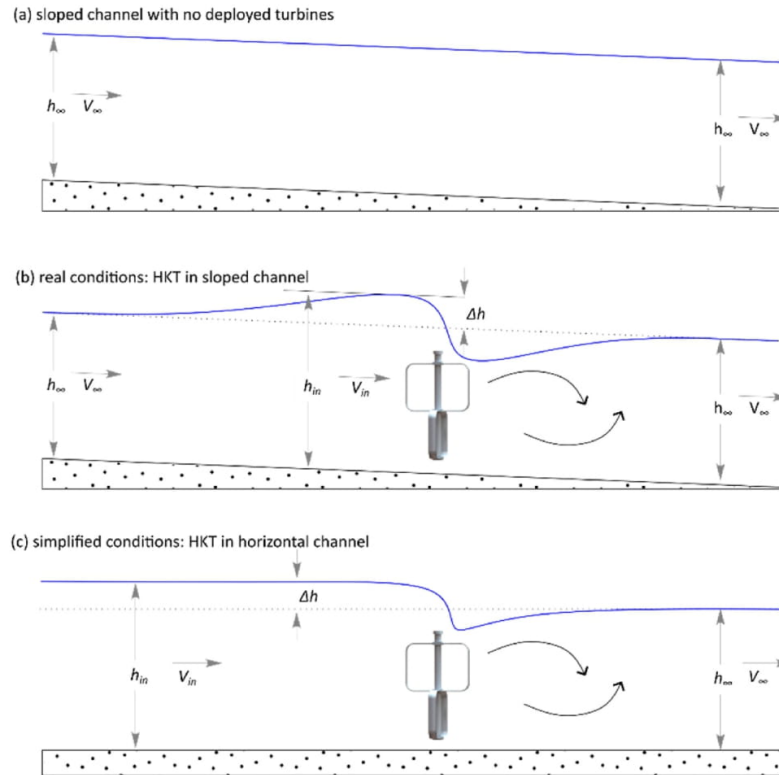


Fig. 5. Schematic representations of (a) sloped bed channel without turbine, (b) with turbine, and (c) horizontal bed channel with a turbine.

algorithm, the CFD settings, and the experimental setup and measurement techniques. Section 4 reports the results of the numerical sensitivity analysis carried out and concludes the final model. Section 5 shows the validation results and reports some critical analysis on the turbine performance for both stand-alone turbine and an array of four turbines. Finally, Section 6 presents the conclusions of the study and some handouts for future developments.

2. Test cases

2.1. Hydrokinetic turbine model

The turbine model (see Fig. 1) used in the present study is an industrial two-bladed H-Darrieus hydrokinetic turbine with a diameter of 2 meters (see Table 1), designed in collaboration with the industrial partner HE-Powergreen in a previous work (Balduzzi et al., 2021). This turbine is segmented at its midspan (1.5 meters), allowing the upper and lower rotors to be positioned perpendicular to each other, with a 90° azimuthal shift. This configuration was implemented to address mechanical balance issues and reduce the output torque ripple. The blades' design uses a proprietary airfoil derived from the DU-06-W-200 by optimizing the thickness distribution and accounting for flow curvature effects (Rainbird et al., 2015). Similar turbine concepts have been employed in various prior studies for different purposes (Mohamed et al., 2022, 2023b; Melani et al., 2021b) at which for all of them, the present ALM code was used, and it showed very good results compared to blade-resolved CFD. (Mohamed et al., 2022), (Mohamed et al., 2023b), (Melani et al., 2021b)

2.2. Channel description

In the current study, both stand-alone turbine and array of four turbines are evaluated. In the case of turbine array, a consistent center-to-center distance of 1.5 times the rotor diameter (1.5D) is used, with the rotational orientation scheme shown in Fig. 2-a. The operational characteristics of the turbines can be found in Table 2.

Fig. 2-b shows the channel used in the present study. It is part of the Biffis artificial channel with a double trapezoidal shape located near the city of Trento, Italy (Il Canale Biffis). The channel section has a width of approximately 21.65 m, with a wetted width of about 20.2 m, corresponding to a volume flow rate of 124 m³/s. The bottom width measures approximately 4.45 m. The maximum height from the bottom to the top is 7.2 m, with a wet height of 6.52 m. More details on the channel are presented in previous work by Cacciali et al. (2023b). In the real channel, the section remains with the same shape with slight variations in the dimensions upstream and downstream the presented section at which the turbine array is installed.

3. Methodology

3.1. Experimental setup

In this section, the setup of the experimental campaign is discussed, including the measurement techniques and the uncertainties associated with the data acquisition and postprocessing.

The performance of the turbines was measured using different types of sensors, some stationary while others rotate, as illustrated in Fig. 3. To avoid signal transmission between fixed and rotating parts using slip rings or wireless transmitters, which are not well-suited for the specific

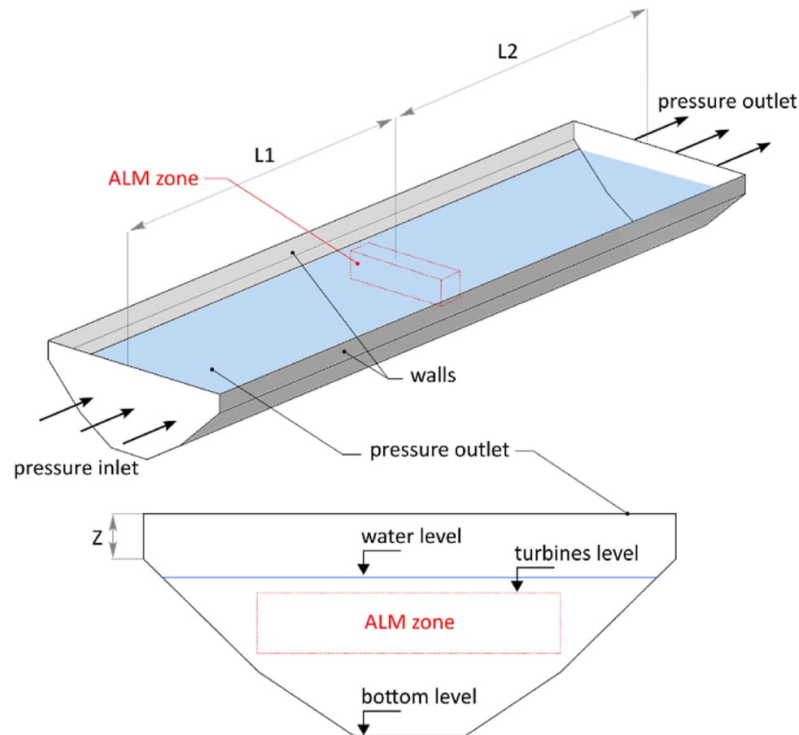


Fig. 6. Schematic representation of the channel CFD domain and boundary conditions.

case at hand, a dedicated data acquisition subsystem was mounted at the upper end of the shaft. This box rotates and is integral with the shaft itself, requiring the shaft to be extended and equipped with an appropriate connection system. The rotating data acquisition subsystem collects and saves signals from sensors installed on the moving parts, which are then exported via physical access, as real-time measurements are not necessary. In parallel with the rotating subsystem, a stationary subsystem was installed to collect signals from transducers attached to the fixed system, synchronized with the former at startup. The residual synchronization error between the rotating and fixed systems is estimated to be approximately ≈ 10 milliseconds.

The sensors connected to the rotating subsystem comprise four strain gauges. Two of these gauges are installed at the midspan of one of the blades on the upper and lower rotors, while the other two are mounted on the spokes connecting the blades of the upper rotor to the shaft. These sensors are responsible for measuring the bending moment with a residual accuracy of about $\pm 2\%$. On the other hand, the fixed subsystem is connected to load cells on the generator support, measuring the total forces acting on the turbine in the X and Y directions with a residual accuracy of $\pm 0.5\%$. Additionally, a single-turn absolute encoder is mounted on the top of the shaft to compute the instantaneous angular position of the blades, with a residual accuracy of $\pm 1\%$.

For velocity measurements, the Acoustic Doppler Current Profiler (ADCP) was utilized. In the present study, the ADCP was mounted on a boat, as shown in Fig. 4-a, with the measurement profile oriented vertically from the free surface to the channel bottom. The instrument used is the WorkHorse-1200kHz by Teledyne-RDI, depicted in Fig. 4-b, which provides a three-components velocity profile with a resolution of 0.25 meters from approximately 0.6 m below the free surface to about 94% of the depth, with maximum depths reaching around 20 m. The velocity standard deviation was limited to less than 1 cm/s by appropriate instrumental settings (Guerrero et al., 2023) for the single

estimation which entailed about 1-Hz acquisition frequency.

As schematically represented in Fig. 4-c, the ADCP consists of four monostatic cylindrical pistons which alternate emission and reception functionalities (namely, monostatic transducers) oriented in a divergent manner, projecting and receiving broadband signals into four acoustic beams from the free surface toward the bottom. The areas not measured near the interfaces mainly depend on the time between emission and reception (i.e., the ringing effect) close to the instrument and the reflection of the side lobes characterizing the beam pattern of cylindrical pistons. This latter limitation is particularly relevant in conditions of channel sloping sides, the profile is limited by the external region of beam pattern (i.e., side lobe) that is reflected by sloping sides more superficially than the beam main axis pointing towards the bottom. The pulses emitted along the four beams are scattered at different depths and then received at successive intervals, determining the vertical resolution of the profile.

To measure the water level variation, a set of water level sensors has been set up at various points along a channel. At each point, there are two types of sensors: IFM – UIT501, which is an ultrasonic level sensor with programmable digital threshold, and Vega – Vegaplug 62, a radar level sensor. Over a period of 48 h, data was continuously collected while four turbines were operational. The process of measuring water levels involves several steps. Initially, raw data is gathered at a rate of 1 Hz from the ultrasonic level sensor at different points. However, a residual offset is present between the points, likely due to slight variations in embankment height and sensitivity to sensor temperature. To mitigate this, a tuning process is implemented, introducing an additional offset to align the average signals across all points along the channel. The corrected water level value is derived using the reference level measured by the Vega radar level sensor to adjust the data across all points. This comprehensive procedure ensures precise water level measurements across the various points, despite inherent challenges and

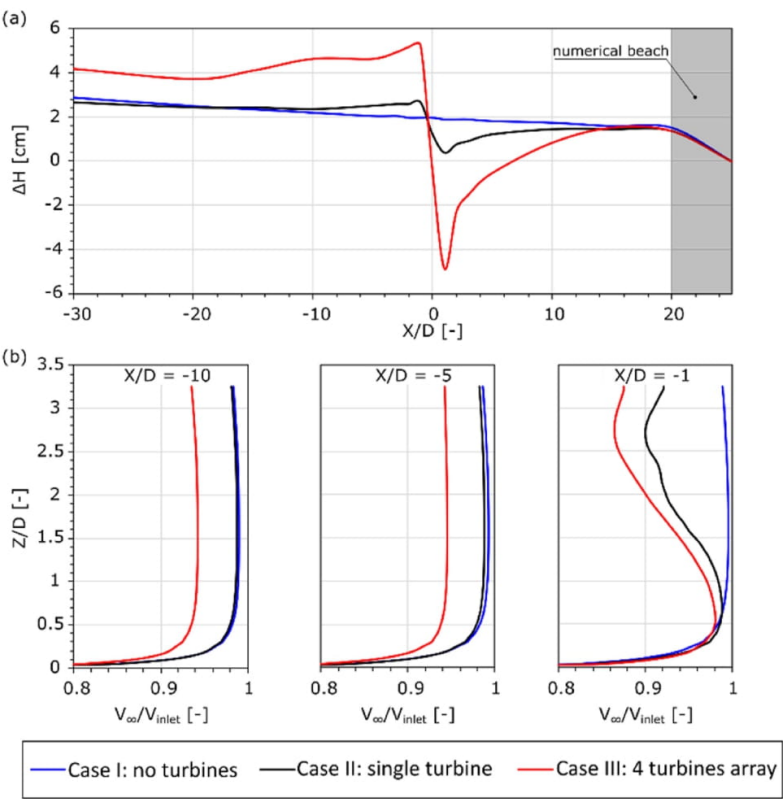


Fig. 7. a) water level variation and b) dimensionless velocity profiles for the channel with no turbines, stand-alone turbine, four-turbine array.

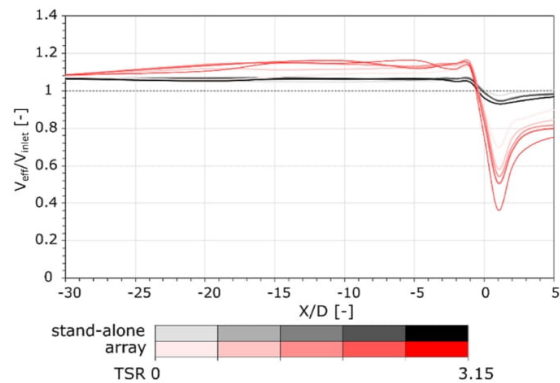


Fig. 8. Variation in the effective velocity V_{eff}/V_{inlet} for the stand-alone turbine and the array at different TSRs.

differences in sensor readings.

3.2. Numerical model

All CFD simulations were conducted using Ansys Fluent 23 R1. This section outlines the key characteristics of the CFD model used, along with a brief overview of the algorithm in the present ALM formulation.

Table 3			
Models used in the domain independency test.			
Name	L1/D	L2/D	Z/D
<i>Effect of height Z</i>			
D1.1	10	25	1
D1.2	10	25	2
<i>Effect of downwind distance L2</i>			
D2.1	10	25	1
D2.2	10	30	1
<i>Effect of upstream distance L1</i>			
D3.1	10	25	1
D3.2	20	25	1
D3.3	30	25	1
D3.4	40	25	1

3.2.1. Actuator Line Method (ALM)

The ALM (Sørensen and Shen, 2002) has recently become a widely used computational tool for simulating wind turbines. The ALM algorithm positions actuator lines representing turbine blades within the computational grid, samples the local Reynolds number and angle of attack for the blade, and utilizes available polar data, along with additional sub-models, to compute aerodynamic forces. These forces are then applied as momentum source terms within the RANS solver to integrate them into the computational domain. In this study, the ALM tool used is a version specifically developed and customized for VATs by some of the authors (Melani et al., 2021a). It is important to note that the current ALM code was specifically designed to be compatible with the Ansys Fluent UDF environment. However, open-source alternatives for ALM in VATs, such as the one developed by Bachant et al. (2018), are also

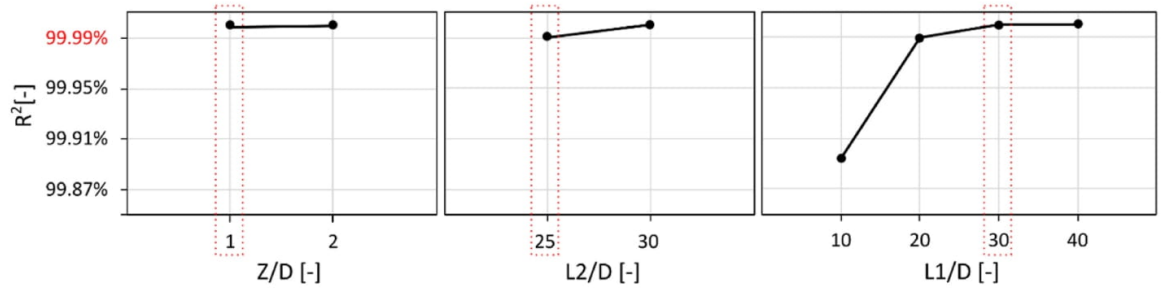


Fig. 9. Domain size sensitivity analysis results: a) effect of height Z , b) downstream distance $L2$, c) upstream distance $L1$.

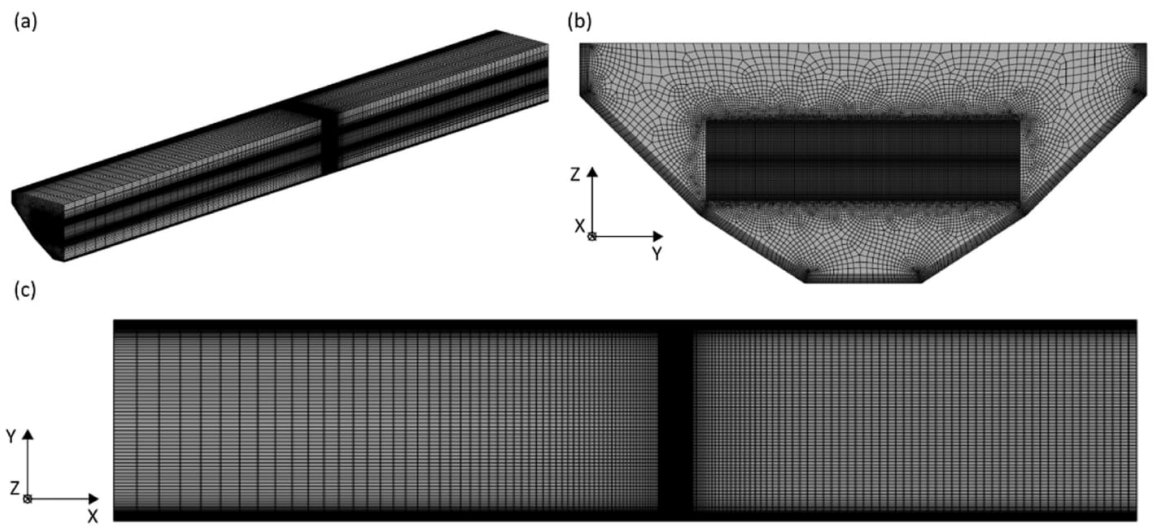


Fig. 10. a) isometric sectional view of the mesh b) sectional front view of the ALM region within the channel c) channel top view.

Table 4

Grids used in the mesh independency test.

Name	#divisions - L1	#divisions - L2	Δh	air-water interface	# elements
M1	60	50	β	n	5,317,444
M2	60	50	$\beta/2$	n	6,625,040
M2B	60	50	$\beta/2$	y	10,824,580
M3	90	75	$\beta/2$	n	8,393,344

available and could be readily used for similar analyses, provided that the coupling with a VOF model is possible. Some unique features of the present code include:

- Sampling of the local flow field:** The LineAverage method, originally proposed by Jost et al. (2018) for horizontal-axis wind turbines and validated by the authors (Melani et al., 2020; Mohamed et al., 2021) for vertical-axis turbines, estimates the blade AoA from the local flow field. It calculates the undisturbed velocity by averaging the velocity field along a circular line with a radius of $r = 1c$, consisting of 80 equally spaced points and centered on the quarter-chord of the airfoil. Additionally, a low-pass filter is applied, implemented using a Cubic Spline Smoothing (CSS) algorithm (de Boor, 1978) following the procedure outlined in (Melani et al., 2021c). (Melani et al., 2020), (Mohamed et al., 2021)

- Polar data:** The quality of the predictions made by the ALM relies heavily on the quality of the polar data it utilizes. This aspect holds particular importance for small Darrieus rotors, as indicated in a prior study conducted by some of the authors (Melani et al., 2021c). In such cases, where low Reynolds numbers and the influence of laminar separation bubbles on airfoil aerodynamics are accentuated, the reliance on high-quality polar data becomes more pronounced. It is noteworthy that the tabulated polar data referenced in the present work pertains to the airfoil that has been virtually transformed, as detailed in (Rainbird et al., 2015). The aerodynamic coefficients before stall were obtained through two-dimensional blade resolved CFD using $k-\omega$ SST with Gamma transport equation to solve transition (Menter et al., 2015). Moreover, the range of the polar data was expanded to encompass the deep stall region, up to an angle of attack of $\pm 180^\circ$, employing the Viterna-Corrigan model (Viterna and Janetzke, 1982), in accordance with the recommendation by Bianchini et al. (Marten et al., 2016) for applications involving Darrieus turbines;
- Forces computation and dynamic stall model:** The present code employs Berg dynamic stall model (Massé, 1981) to account for the unsteady flow effects on blade loads, as elaborated in (Melani et al., 2021c). This model introduces a delay in the measured angle of attack, which varies proportionally with the flow reduced frequency, thereby incorporating dynamic effects into the solution. Lift and drag coefficients are determined through linear interpolation between

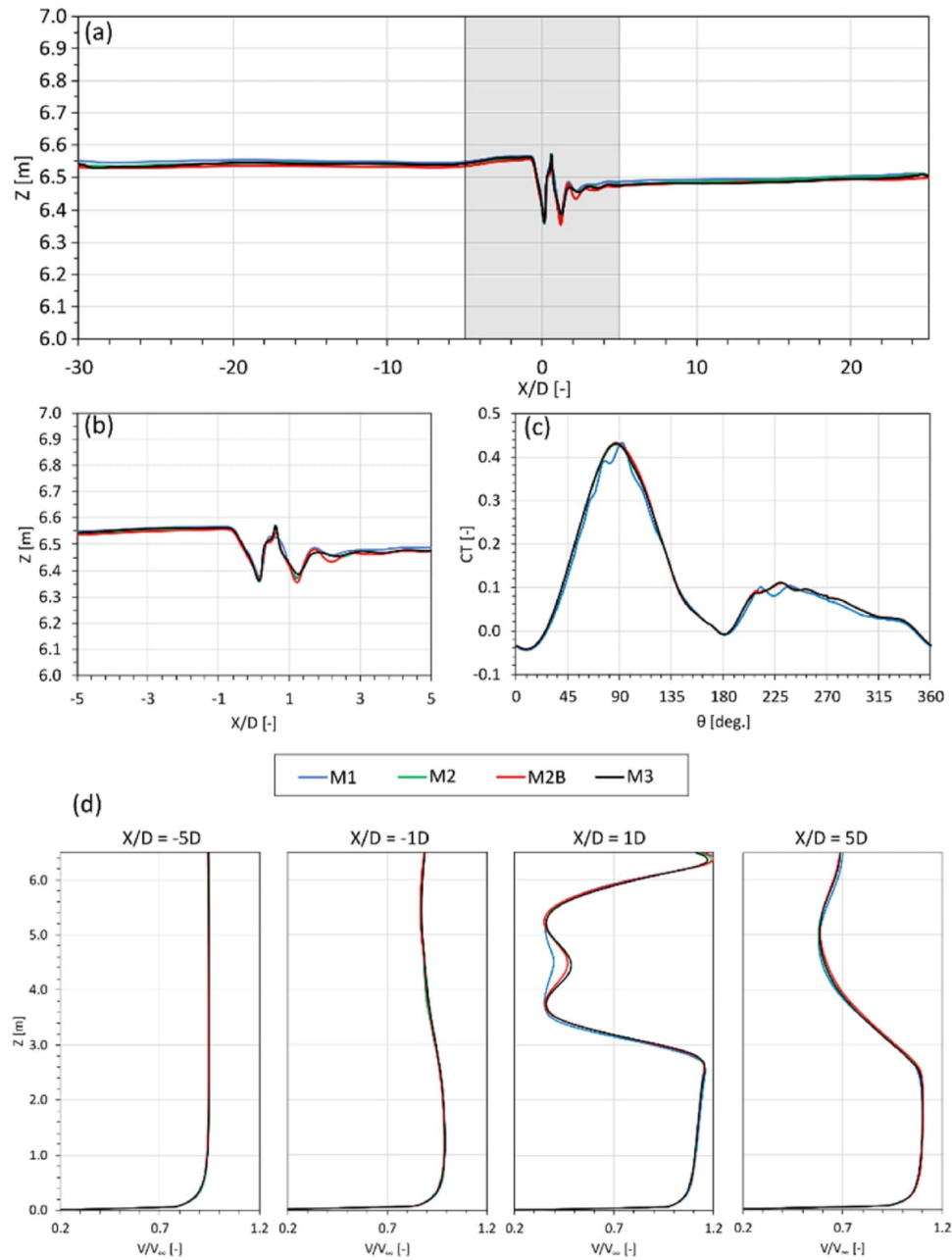


Fig. 11. Results of the mesh independency test a) water level along the full channel with a zoomed area of b) water level near the turbines c) instantaneous torque coefficient d) dimensionless velocity profiles.

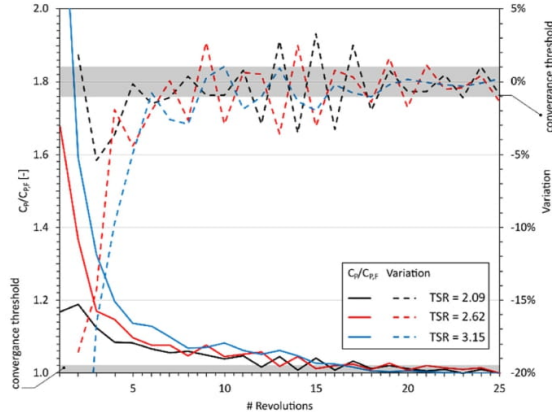


Fig. 12. Convergence histories based on the power coefficient at TSRs 2.09, 2.62, and 3.15.

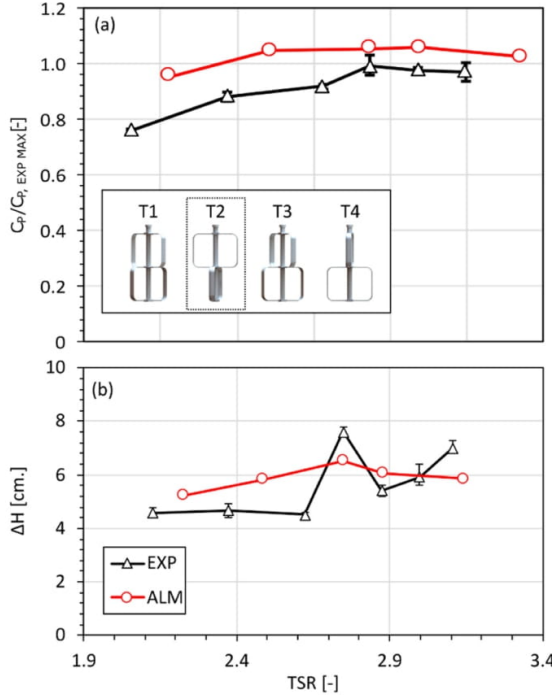


Fig. 13. Comparison between average power coefficient obtained numerically and that of the experimental data normalized over the maximum experimental C_p for turbine 2 in the four-turbine array.

static and dynamic coefficients, with a weight given by the constant A_M , here set to 6. Added mass terms were also included by modifying the normal and tangential forces coefficients, $C_{N, AM}$ and $C_{T, AM}$, according to Eqs. 1 and (2):

$$C_{N, AM} = -\pi \cdot c \cdot \frac{dV_N}{dt} \frac{1}{8 V_{rel}^2} \quad (1)$$

$$C_{T, AM} = \pi \cdot c \cdot \frac{d\alpha}{dt} V_N \frac{1}{8 V_{rel}^2} \quad (2)$$

with c the chord length, V_{rel} the relative velocity, and V_N the normal velocity. dV_N/dt and $d\alpha/dt$ are the rate of change of the normal velocity and angle of attack, respectively.

- **Tip effects correction:** The Dag-Sørensen (DS) correction (Sørensen et al., 2014) is applied in the present work to account for tip effects. The application of this correction to the ALM is based on the similarities between the effects on the flow field caused by the ALM's isotropic Gaussian smearing function and those of a Lamb-Oseen vortex, with a viscous radius equivalent to the kernel width β . Melani et al. (2022) demonstrated and successfully applied the correction to a Darrieus turbine, showing reasonably accurate results. It is noteworthy that the current tip effects correction formulation does not encompass the contribution of the vorticity shed in the wake, unlike the original work (Sørensen et al., 2014). Additionally, the application of this correction to blades with curved ends, such as the one under investigation, might introduce some uncertainties. Although the authors have recently made efforts to tune the ALM for simulating tip effects (Melani et al., 2024a, 2024b), their effects on rotating blades are more complex and warrant a separate investigation. (Melani et al., 2024a), (Melani et al., 2024b)

Additional information about the present ALM code formulation can be found in (Melani et al., 2021a).

3.2.2. CFD setup

In order to build a CFD model that is capable of simulating the physical parameters of interest, careful consideration must be taken into account while putting the assumptions that should reduce the computational cost. As illustrated in Fig. 5-a, for a sloped uniform channel, i.e., constant cross-section, a steady equilibrium must be satisfied between the gravity force associated to the slope of the channel and the friction on its walls. Thus, the water depth h_∞ and flow velocity V_∞ at the inlet remain constant as long as the uniformity and steadiness constraints are maintained. Deploying HKTs into that system (Fig. 5-b) would perturb the water level, altering in turn the stream velocity upstream of the turbine. Based on the energy balance, the blockage effect due to the turbine will cause the water level to increase upstream to a maximum height of h_{in} while decreasing the flow velocity to V_{in} . Downstream of the turbine, the water level decreases again, for then returning to its unperturbed value h_∞ far downstream. This is also known as “depth recuperation”. The upstream rise in water level corresponds to the drop in hydraulic head, $\Delta h = h_{in} - h_\infty$ that is a feasible assumption for negligible head dissipation.

Resolving such system using CFD would require extremely long domains, as reported by Gauvin-Tremblay and Dumas (Gauvin-Tremblay and Dumas, 2020). Alternatively, a simplified model can be created, in which the channel bed can be assumed horizontal (“horizontal bed assumption”), thus producing a water level profile like that illustrated at Fig. 5-c. In these conditions, the VOF (Hirt and Nichols, 1981) method allows the water level to vary based on the resolved flow field, fixing it at the outlet while allowing it to change at the inlet. The main drawback of this assumption is that it is not possible to compute distance required to reach the “depth recuperation”. However, the estimation of the upstream rise, Δh , is still feasible.

Regarding the VOF method, it was employed to simulate the channel free surface, solving the flow fields of water and air through a single set of momentum equations while tracking the volume fraction of each fluid across the domain. This method operates on the principle that two or more fluids cannot compenetrates. It introduces the volume fraction of each fluid as a new variable for every computational cell. The volume fraction equation is then solved for the secondary phase, while that of the primary phase is computed so that the total volume fraction sum is unity (ANSYS FLUENT 12). A constant surface tension of 0.072 N/m was set to the water surface. Regarding the numerical discretization of the

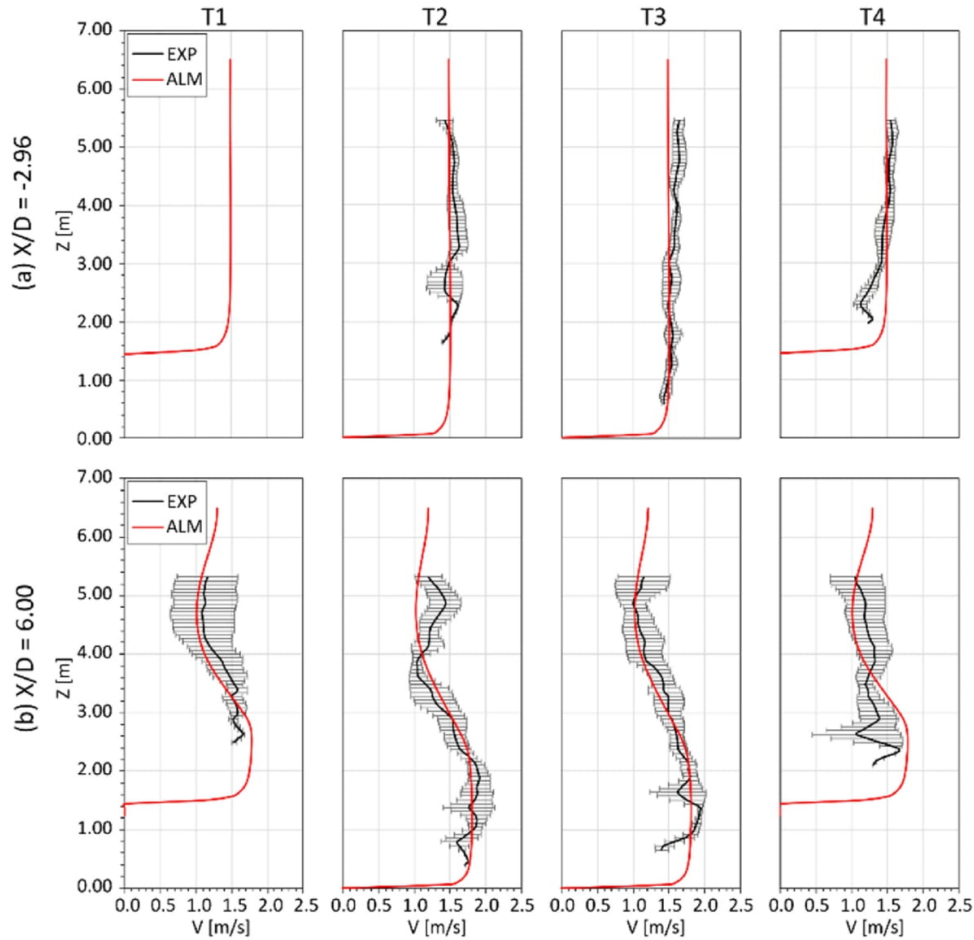


Fig. 14. Comparison between numerical model and experimental data velocity profiles for the four-turbine array upstream and downstream the turbines.

volume fraction, two different schemes were evaluated: compressive and modified High Resolution Interface Capturing (HRIC). However, both schemes predicted the same variations in the water level, so the default compressive scheme was used.

Three-dimensional simulations were performed using the unsteady Reynolds-Averaged Navier-Stokes (URANS) solver. The $k-\omega$ Shear Stress Transport (SST) model (Menter, 1993) was selected for the turbulence closure, renowned for its accuracy and stability in unsteady flow conditions (Balduzzi et al., 2016a). The pressure-based solution was employed using a coupled algorithm, and a second-order upwind scheme was utilized for the spatial discretization of all transport equations. The time difference employed a second-order implicit scheme. Additionally, based on previous experience on this type of simulations, the residuals for all variables were set to 10^{-5} , and 20 inner iterations were performed per time step, where the time step size Δt corresponds to $\Delta\theta = 1^\circ$, based on the sensitivity analysis performed by Melani (Melani, 2018). Furthermore, other simulation details, including the domain and grid sizes were selected based on a dedicated sensitivity analysis that is presented in Section 4.

Fig. 6 shows the computational domain and boundary conditions of the present model. The domain includes the channel cross-section extruded in both upstream (L1) and downstream (L2) directions, as well as a volume of air of height Z on top of the water surface. Its

dimensions were selected to ensure the consistency of the applied boundary conditions (see Section 4.2): i) pressure inlet at the inlet, with a prescribed velocity used to compute the dynamic component of pressure, treated as the total pressure prior to solving the flow field due to the absence of height differences, thanks to the horizontal bed assumption; ii) pressure outlet at the outlet of the channel and the top boundary; iii) no-slip condition at all channel walls. It is important to note that, for a simplified horizontal open channel model, the solution is initialized assuming a flat water surface, with constant velocity from inlet, then boundary conditions set at the inlet change as the simulation run, due to the variation in the energy gradient level along the channel, causing the pressure field to change thanks to the flow interactions with the turbines and the channel walls, which in turn alters the velocity and water level upstream compared to its prescribed values at the pressure inlet. Finally, although no waves have been imposed at the inlet, a numerical beach is applied before the outlet to create an artificial wave-damping procedure to avoid flow reversal.

4. CFD sensitivity analysis

This section presents the main results of the sensitivity analysis carried out on the proposed CFD model, initially highlighting the significance of the assumption of having a horizontal bed instead of sloped

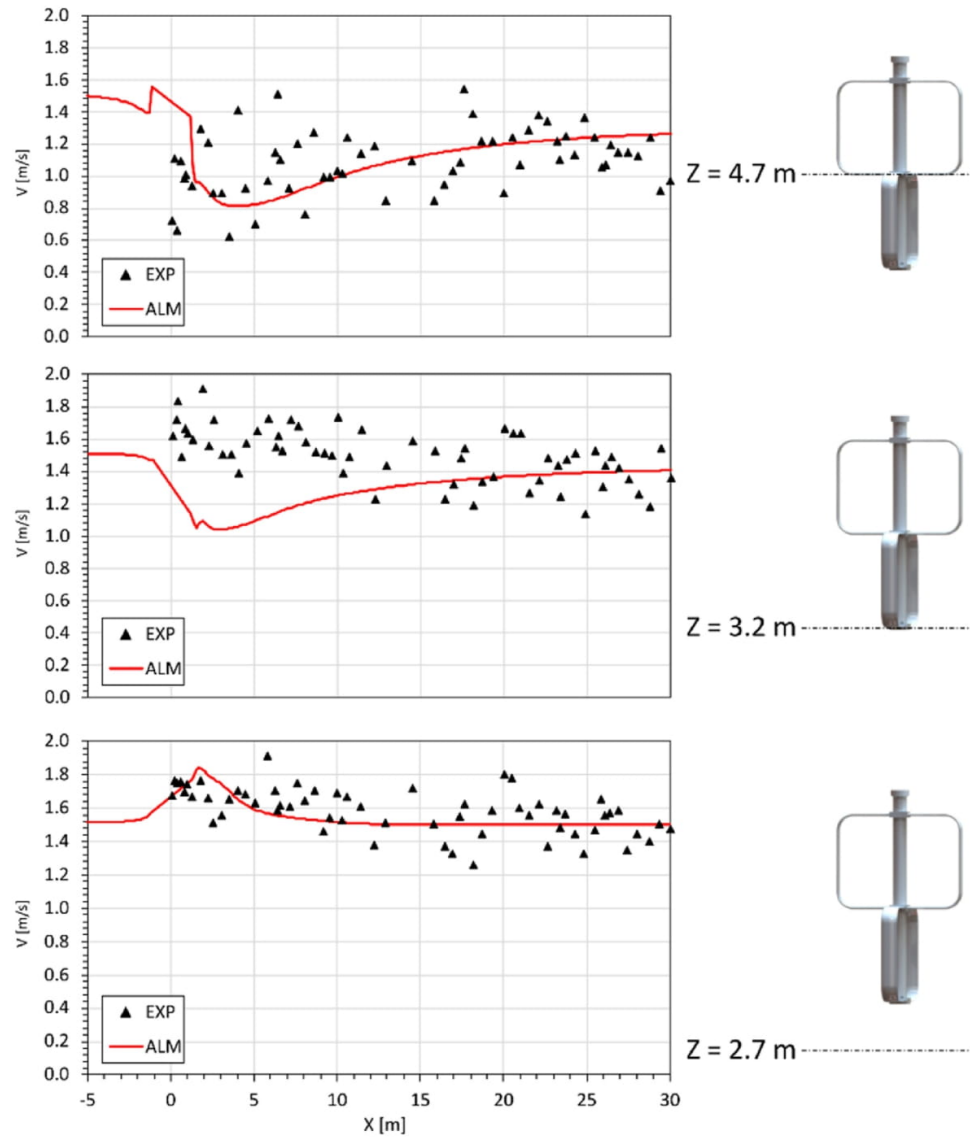


Fig. 15. Comparison between numerical model and experimental measurements of velocity in the wake of T2 centre for a four-turbine array at different Z levels.

one (see Section 4.1), then moving to the numerical independence analysis for the domain (see Section 4.2) and mesh sizing (see Section 4.3).

4.1. Testing the horizontal bed assumption

To assess the horizontal bed assumption, three different cases were simulated: Case I, where no turbines are operating inside the channel, Case II with a single operating turbine located at the channel center, and Case III, where an array of four turbines is operating, with the layout mentioned previously in Fig. 2-a. Fig. 7 shows a comparison between the water level variation in terms of ΔH and the velocity profile at different upstream distances for the three cases. It is important to highlight that the water level was computed for several 2D cross-sections across the domain length, thus accounting for the three-dimensional nature of the

channel. As can be observed, the water level in Case I shows a smooth slope from inlet to outlet with no perturbations throughout the domain; given that the channel is assumed to have a horizontal bed, this is due to the friction losses. In Case II, the water level follows that of Case I far upstream, while the effect of blockage due to the interaction between the water stream resource and the operating turbine can be observed at $X/D < -10$. On the other hand, for Case III, the blockage effect is much stronger than that in Case II due to the presence of four turbines, which results in a significant upstream rise that is propagated up to the inlet. For all the cases, the impact of the numerical beach set at outlet is clearly visible, damping the induced waves at outlet to prevent backward flows.

The efficiency or the power coefficient C_p of the turbines is computed through dividing the power generated P over that available in the water resource, which could be written as:

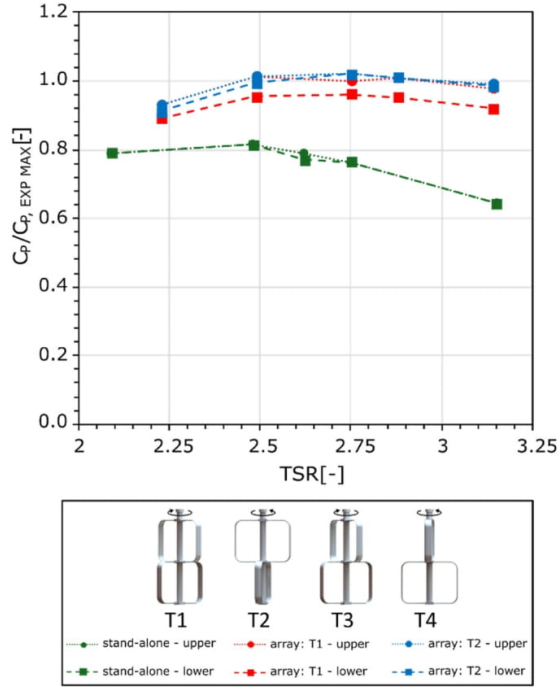


Fig. 16. Comparison between the average power coefficient normalized over the maximum experimental C_p for each rotor of the HKT design for the stand-alone turbine and the array.

$$C_p = \frac{P}{0.5\rho AV_\infty^3} \quad (3)$$

Where ρ is the water density, A is the turbine swept area, and V_∞ the freestream velocity. Concerning the latter, it can be noticed that the blockage affects the inflow velocity profiles as in Fig. 7-b, in which Cases II and III show slowdown in flow velocity compared to Case I, which is more pronounced at a distance $1D$ upstream. This decrease in the inflow velocity before the turbines should be considered while computing the power coefficient of the turbines C_p , otherwise the computed C_p would have an artificial increase that is not independent from Froude Number, as analyzed in detail in (Gauvin-Tremblay and Dumas, 2020). To this end, one can describe the flow at any channel cross-section plane in terms of its specific energy E :

$$E = \Delta h + \frac{V^2}{2g} \quad (4)$$

where Δh is the difference in the water surface altitude, V is the flow velocity, and g is the gravitational acceleration due to gravity, which was added to the model in the $-Z$ direction. Thus, given the horizontal bed assumption, the initial boundary conditions at inlet could be written as $E_{inlet} = V_{inlet}^2/2g$, and a new parameter can be introduced to represent the effective flow velocity V_{eff} at any cross-section plan across the domain:

$$V_{eff} = \sqrt{2gE} \quad (5)$$

Fig. 8 reports the ratio between the effective flow velocity and the inlet velocity initially prescribed at inlet V_{eff}/V_{inlet} for both stand-alone turbine and an array of four turbines at different Tip-Speed Ratios (TSRs), the TSR is defined as the velocity of the turbine to the velocity of the inflow stream:

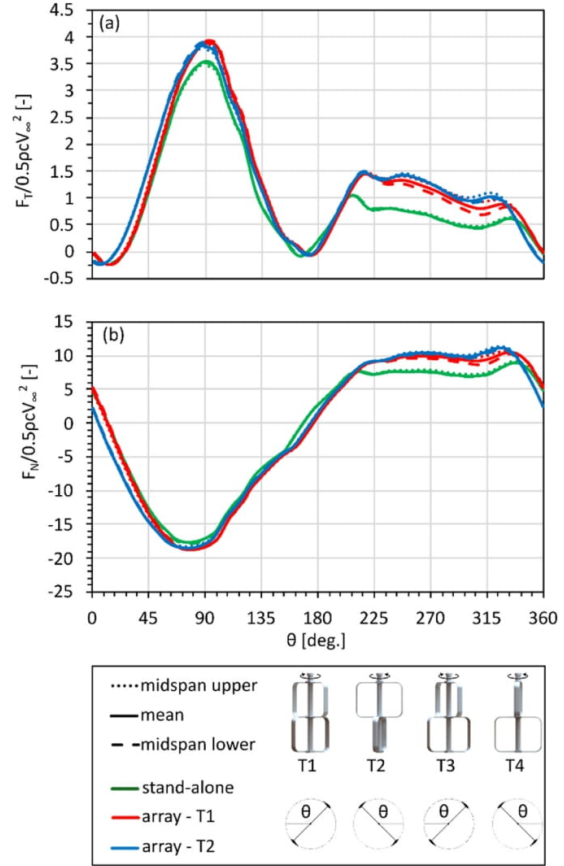


Fig. 17. Non dimensional (a) tangential and (b) normal forces on a single blade of the stand-alone turbine and the array.

$$TSR = \frac{\omega R}{V_\infty} \quad (6)$$

As can be seen, the specific energy upstream the turbines is higher than the one specified at inlet, then the energy decreases due to the power extracted from the turbine. It can be also noted that the total specific energy upstream is slightly above the one specified initially at inlet; this excess energy is indeed the effect of blockage, which should not be taken in consideration while computing the power coefficient. Thus, the velocity seen upstream the turbines, which is often less than the one defined at inlet, due to the presence of Δh , cannot be taken as a reference velocity to normalize the power coefficient as it does not represent the total specific energy available. Hence, V_{inlet} prescribed at inlet was considered to compute the C_p .

4.2. Domain size

In the domain sensitivity analysis, the goal was to investigate how varying the distances $L1$ and $L2$, as well as the height Z , affects both the performance of the turbines and the behaviour of the flow field. A total of eight different domains (detailed in Table 3), obtained by incrementally adjusting the values of $L1$, $L2$, and Z individually, were tested. All dimensions are normalized over the rotor diameter, D , for generality.

The coefficient of determination R^2 (see Eq. (7)) for the instantaneous torque, $C_T(\theta)$ was used to evaluate the independence of the

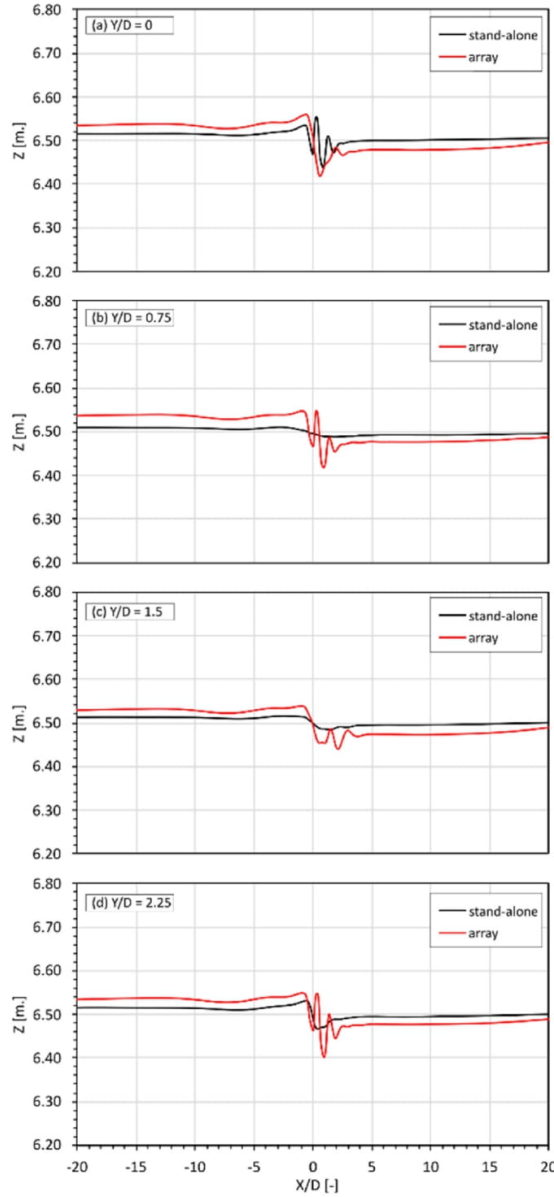


Fig. 18. Variation in water level at different lateral distances, comparison between stand-alone and array.

numerical solution with respect to a reference setup (subscript “final”) (Balduzzi et al., 2016b), here identified with domains D1.2, D2.2, and D3.4 for the variables Z , $L1$, and $L2$, respectively.

$$R^2 = 1 - \frac{\sum_{\theta=0^\circ}^{360^\circ} [C_T(\theta) - C_{T,final}(\theta)]^2}{\sum_{\theta=0^\circ}^{360^\circ} [C_T(\theta) - C_{T,average}]^2} \quad (7)$$

Fig. 9 reports the R^2 results for the selected domains. Altering the height Z or the downstream distance $L2$ has no significant impact on the resulting torque considering a threshold of $R^2 > 99.99\%$. However, variations in $L1$ notably influence the turbine performance. It is important to note that, in the present simulation, and unlike the

simulations that assume open field, the flow development upstream of the turbines, including the free surface rise, plays an important role in estimating the inflow conditions near upstream the turbines. Thus, sufficient upstream distance is required to guarantee the upstream flow fully development. Based on the results presented in this section, D3.3 was selected for the rest of the simulations.

4.3. Mesh independency test

A dedicated mesh sensitivity analysis was carried out to achieve a proper spatial discretization, aiming at the best compromise between accuracy in capturing spatial flow variations and computational efficiency. As shown in Fig. 10, the domain is discretized using a combination of hexahedral and tetrahedral mesh. The ALM region is discretized with structured mesh of uniform element size Δh . The streamwise distance as well as the height of the ALM region are divided into 30 divisions. Furthermore, 15 inflation layers were added near the channel walls to ensure that $y^+ < 1$. Table 4 displays the different grids that were evaluated. The analysis focused on three main parameters: (i) ALM element size Δh in terms of kernel width β , which was fixed at $0.25c$ for all cases; (ii) meshing strategy at the air-water interface, ensuring uniform hexahedral elements in this area; (iii) the number of divisions along $L1$ and $L2$.

Fig. 11 illustrates the results of the mesh sensitivity analysis concerning water level, torque coefficient, and dimensionless velocity profiles. Initially, as depicted in Fig. 10-a), it is evident that, except for M1, the channel water level matches at the inlet and outlet for almost all other grids, with an upstream rise of approximately 0.04 m. Fig. 10-b) provides a more detailed profile of the water level within $-5 < X/D < 5$. Notably, almost no significant difference can be observed between case M2B and the other cases, suggesting the unnecessary use of structured mesh in the air-water interface for this case. The cyclic torque coefficient variation is shown in Fig. 10-c) for the evaluated meshes, indicating identical torque variation for all cases except M1, which exhibits some volatile variation around $\theta \approx 90^\circ$ and 225° . In Fig. 11, velocity profiles are depicted at the center of the channel for different distances upstream and downstream of the turbines. All grids exhibit similar velocity profiles upstream of the turbines. However, differences can only be noticed between M1 and the other grids, particularly at $X/D = 1$, highlighting the relative importance of properly sizing the ALM region. Nonetheless, the overall results are consistent for M2, M2B, and M3. Therefore, based on the above results, the mesh settings adopted for M2 were chosen for the rest of the simulations.

4.4. Convergence criteria

To ensure that the final solution is independent on the number of simulated revolutions, a dedicated analysis was carried out. As suggested in (Balduzzi et al., 2016a), two criteria were assessed: the first was based on the cyclic average power coefficient normalized by the final cycle's average power coefficient, while the second was based on the variation between the power coefficient of each cycle and that of the subsequent cycle. Based on literature recommendations and authors' experience, the convergence thresholds were set at 1.02 and $\pm 1\%$, respectively.

As shown in Fig. 12, the TSR plays a significant role during the initial phase of the simulation. At higher TSRs, initial cycles show greater divergence from the final solution. This divergence is linked to wake development. Conversely, for the same number of revolutions a lower TSR results in a longer simulated time, granting more time to the near wake to be convected downstream. Based on the analysis of Figs. 12 and 20 revolutions were found to be sufficient to reach the final solution, and this number was applied in all subsequent simulations.

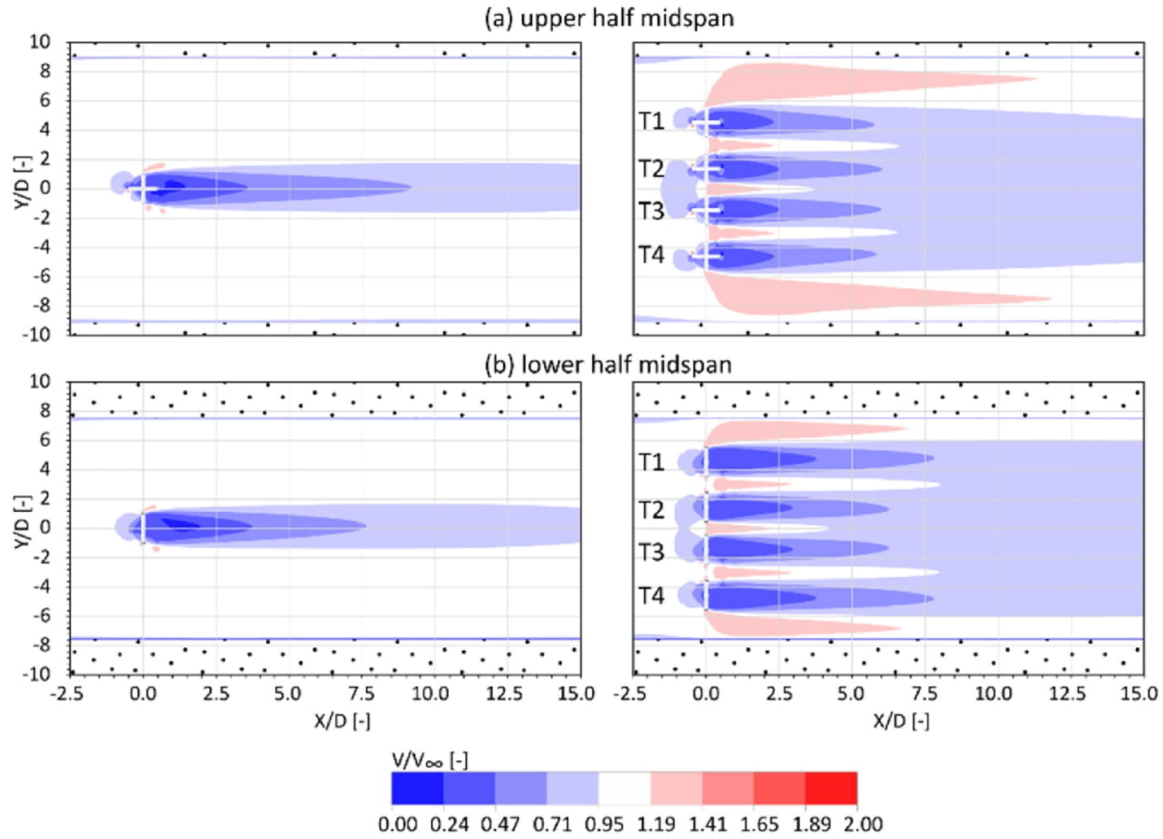


Fig. 19. a) dimensionless contours of the stand-alone turbine and the array at $TSR = 2.75$ plan view at mid-plane of lower and upper rotor.

5. Results and discussion

In this section, the main results obtained by the proposed numerical model are discussed. Initially, the validity of the model has been assessed by comparing its predictions against experimental measurements (see Section 5.1). Then, the performance of the HKT array is compared in Section 5.2 against the stand-alone turbine to assess the impact of deploying Darrieus turbines in close proximity in terms of efficiency improvement, as well as the influence on the channel itself and the flow field.

5.1. Validation

Fig. 13-a reports the experimental results of the average power coefficient, C_p of T2 within the four-turbines array at various TSRs normalized over the maximum experimental C_p for T2 (which occurs at $TSR = 3.15$), compared with that obtained from the present numerical model. C_p normalization is due to industrial non-disclosure constraints, although the normalization does not alter the comparative analyses presented in the study. In general, very good agreement is found between the numerical and experimental results at $TSR \geq 2.75$, while at low TSRs, the numerical model shows overestimated power coefficient. Regarding the observed upstream rise reported in Fig. 13-b, it can be noticed that the experimental results show that ΔH is almost independent from the turbine rotational speed up to $TSR \approx 2.5$. Thereafter, ΔH increases to its maximum value at $TSR = 2.75$, corresponds also to the maximum C_p , followed by a decrease and then an increase again. It can

be noticed that the ΔH increase noticed at $TSR > 2.75$ corresponds to an increase also in the C_p . This relation between the extracted power and the height of the upstream rise is not straightforward in the experimental results, as measurements at low TSRs does not show that. This might be attributed to uncertainties related to slight variations in the far upstream water level, which was not perfectly stationary during the measurement hours. However, the numerical model predictions showed a relation between the extracted power and ΔH , showing a degree of proportionality between the observed upstream rise height and the computed C_p .

To validate the numerical model in terms of reproducing the water velocity flow field, Fig. 14-a depicts time-averaged vertical velocity profiles measured upstream of each turbine at $X/D = -2.96$ compared to the CFD results. It is evident that the inflow reproduced using ALM remains almost constant across the entire cross-sectional area, while experimental measurements exhibit some variations in velocity values. However, when comparing the velocity profiles, both CFD and experimental measurements demonstrate acceptable matching, indicating similarity in the inflow conditions. Some discrepancies are noticeable near the wall, which were attributed to the wall conditions used in the numerical simulation and the uncertainties in the experimental measurements near the wall.

In Fig. 14-b, velocity profiles sampled downstream of each turbine resulting from the ALM simulation are compared with those reconstructed using experimental measurements at a downstream distance of $X/D = 6$. Overall, a velocity deficit can be observed downstream of the turbines, with some acceleration regions appearing on the sides and at the channel bottom due to the blockage effect when the flow field

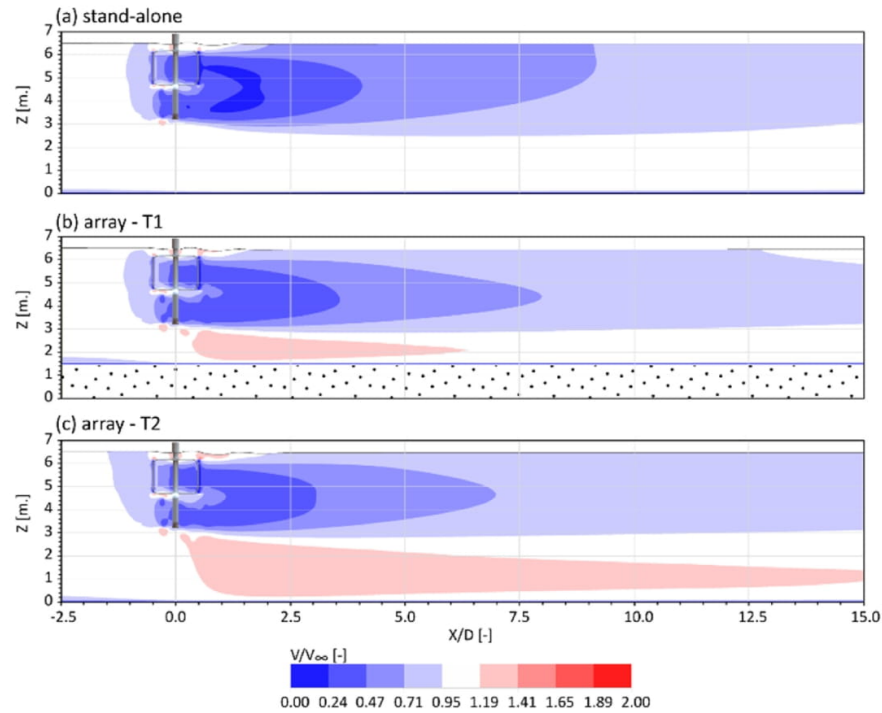


Fig. 20. Elevation view of the dimensionless velocity contours for the stand-alone compared with b) T1 and c) T2 in the array.

around the turbines interacts with the channel walls. The velocity profiles downstream of the turbines provide an indication of both the overall performance of the ALM and the effectiveness of its integration within a VOF solution. A general similarity is found in the shape of the velocity profiles. Discrepancies between the experimental and CFD results are noticeable near the walls, where uncertainties associated with the use of wall functions, in addition to experimental uncertainties in measurements near the wall, might be relevant.

Fig. 15 presents a comparison between the numerical model and the experimental measurements of velocity downstream from the center of T2 at various Z levels. Despite the significant variability observed in the experimental data, a discernible trend is evident. Overall, good agreement is achieved for $X > 10$ m. However, the near wake region is not accurately captured for the strut lines at $Z = 6.2$ m and 3.2 m, likely due to the unmodeled struts and the approximated tip-effect modeling, which ignores the curved blades at tips as detailed in section 3.2.1. Notably, there is a very strong agreement along the measurement line under the turbines ($Z = 2.7$ m), particularly in the acceleration region near the turbines.

5.2. HKT array performance

Moving to the analysis of the operational characteristics of the HKT design presented in this work along with the proposed array arrangement, Fig. 16 shows the average power coefficient for each turbine half in the array compared to the stand-alone turbine, normalized over the maximum experimental C_p for T2. Due to the symmetry in turbine performance, T3 and T4 produced identical results to T2 and T1, respectively, so only the performance of T1 and T2 are presented.

In general, comparing the stand-alone turbine to the array shows that the efficiency of the array can be improved by up to 25%, with a shift in the optimum TSR from approximately 2.5 for the stand-alone turbine to 2.75 for the array. It is important to note that the performance curve of

the array is almost flat between TSR values of 2.5 and 3.1. Additionally, for the turbines on the sides (T1 and T4), the upper half produces more power than the lower half. In contrast, the turbines in the middle of the array (T2 and T3) produce almost the same power in both halves, with only a small difference noticeable at low TSRs. This difference in C_p between the lower and upper halves of the side turbines is attributed to the trapezoidal shape of the channel, suggesting that the lower rotor experiences more flow interactions with the channel walls.

Fig. 17 shows the normalized instantaneous tangential F_T and normal F_N forces over one revolution for a single blade of T1 and T2 compared to that of the stand-alone turbine. For T1, where the mutual interaction is only with the adjacent T2 in the inward half, the tangential force is similar to the stand-alone turbine during the first quarter ($0^\circ < \theta < 90^\circ$), with a higher peak at $\theta \approx 90^\circ$. During the second quarter ($90^\circ < \theta < 180^\circ$), T1 shows a significant increase in tangential force compared to the stand-alone turbine due to the interaction with T2. An increase in the tangential force is also noted during the downstream half ($180^\circ < \theta < 360^\circ$). The differences in tangential force between the upper and lower halves are more pronounced during the downstream half of the revolution.

Similar observations are made for the normal force on T1. The normal force closely matches that of the stand-alone turbine during the first quarter, with no mutual interaction with other turbines. However, during the second quarter, the normal force of T1 is significantly higher than that of the stand-alone turbine, and the differences between the upper and lower halves are more pronounced during the downwind half of the revolution.

On the other hand, T2 exhibited higher tangential and normal forces on both the windward and leeward sides compared to the stand-alone turbine, due to its mutual interaction with T1 and T3. Throughout the entire downwind half, the forces on the turbine blade were of greater magnitude, slightly exceeding those of T1 at most downstream azimuth positions.

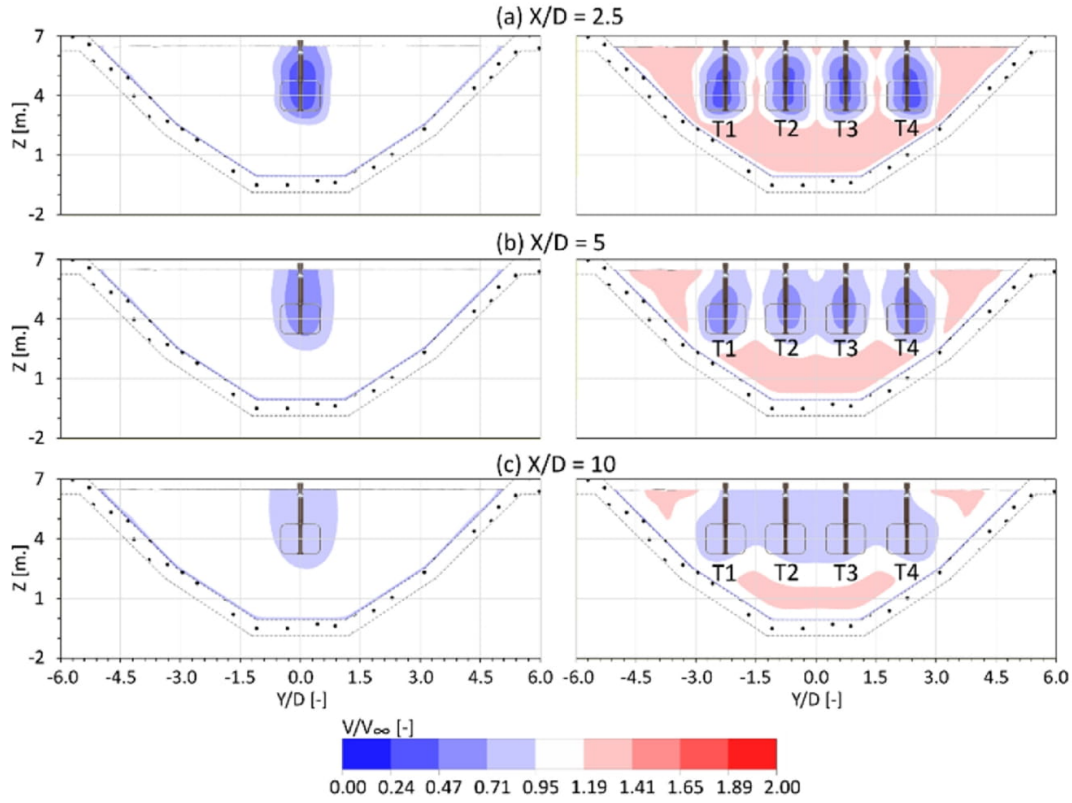


Fig. 21. Front view of the dimensionless velocity contours for the a) stand-alone turbine, b) T1 and c) T2 in the array.

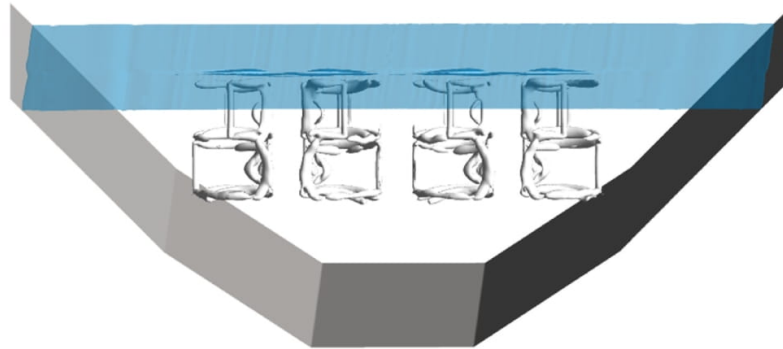


Fig. 22. Two iso surfaces for λ_2 criterion showing the vortex cores resulting from the four-turbine array, and the free water surface.

5.3. Hydraulic impact on the channel

To investigate the impact of deploying HKTs on the channel itself, instantaneous water level variations after the 20th cycle are reported for both the stand-alone turbine and the array case at various Y/D distances, as shown in Fig. 18. Since the solution is symmetrical around the channel vertical longitudinal mid-plane, only positive Y/D values are reported, with the corresponding negative values being equal. As seen, deploying the array results in a significantly higher upwind rise across all Y/D sections. In contrast, the blockage induced by the stand-alone turbine is not sufficient to significantly raise the water level,

especially on the sides, far from the turbine located at the channel centre. For the stand-alone turbine, the average upstream rise along the canal width is approximately 0.06 m, peaking at about 0.12 m at the centre. On the other hand, the average upstream rise for the array is 0.13 m, with minimal deviation from the peak value at the channel centre, which is 0.14 m.

Fig. 19 shows the velocity contour, made non-dimensional by the inlet velocity V_∞ , in the top view for the mid-plane of both the lower and upper turbine of the stand-alone turbine and the four-turbine array. The blockage effect is evident in the array case, with accelerated flow observed between the turbines themselves as well as between the

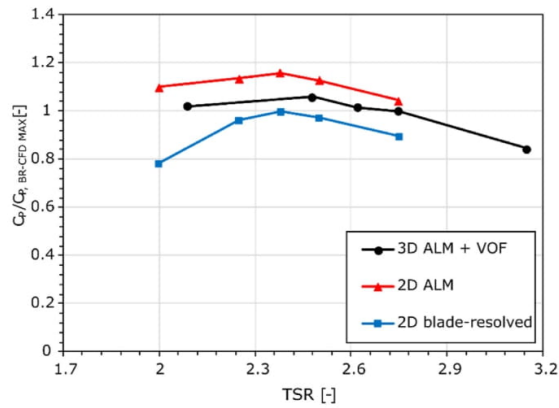


Fig. 23. Comparison between the average power coefficient C_p normalized over maximum C_p of the blade-resolved CFD for the stand-alone turbine using 2D ALM, 3D ALM and blade-resolved CFD.

turbines and the channel wall on the sides. The difference in accelerated flow between the upper and lower halves of the array is significant; the upper half experiences more intense acceleration zones compared to the

lower half. It is important to note that the lower parts of the turbines are subjected to a smaller cross-sectional channel area due to the trapezoidal shape. However, this difference is not sufficient to cause a significant impact on the wake dissipation mechanism or the power coefficient.

Fig. 20 shows the elevation view of the dimensionless velocity contour for the stand-alone turbine compared with T1 and T2 within the array. Slightly faster wake dissipation can be observed in the array case compared to the stand-alone turbine, particularly near the free water surface at $X/D > 2.5$. More importantly, an acceleration zone forms below the turbines in the array, with velocities reaching about $1.2V_\infty$. This acceleration zone is due to flow blockage, where the inflow anticipated by the turbines is bypassed below the array, causing significant acceleration, which is not present in the stand-alone turbine case.

Fig. 21 shows the dimensionless velocity contours from the front view at several channel cross-sections for $X/D = 2.5, 5$, and 10 . It can be seen that this acceleration zone remains around the turbines up to $X/D = 10$, after which the wake gradually returns to V_∞ . Another observation from these contours is the shape of the wake produced by this design, which has two connected parts. This enhances the mixing of the wake and does not exhibit the typical skewed wake shape known for VATs (see Fig. 22).

From the analysis, it is evident that deploying an array of HKTs increases the local acceleration zones in the channel near the turbines, significantly higher than the stand-alone turbine due to the imposed blockage, particularly between the turbines and the channel bed in the

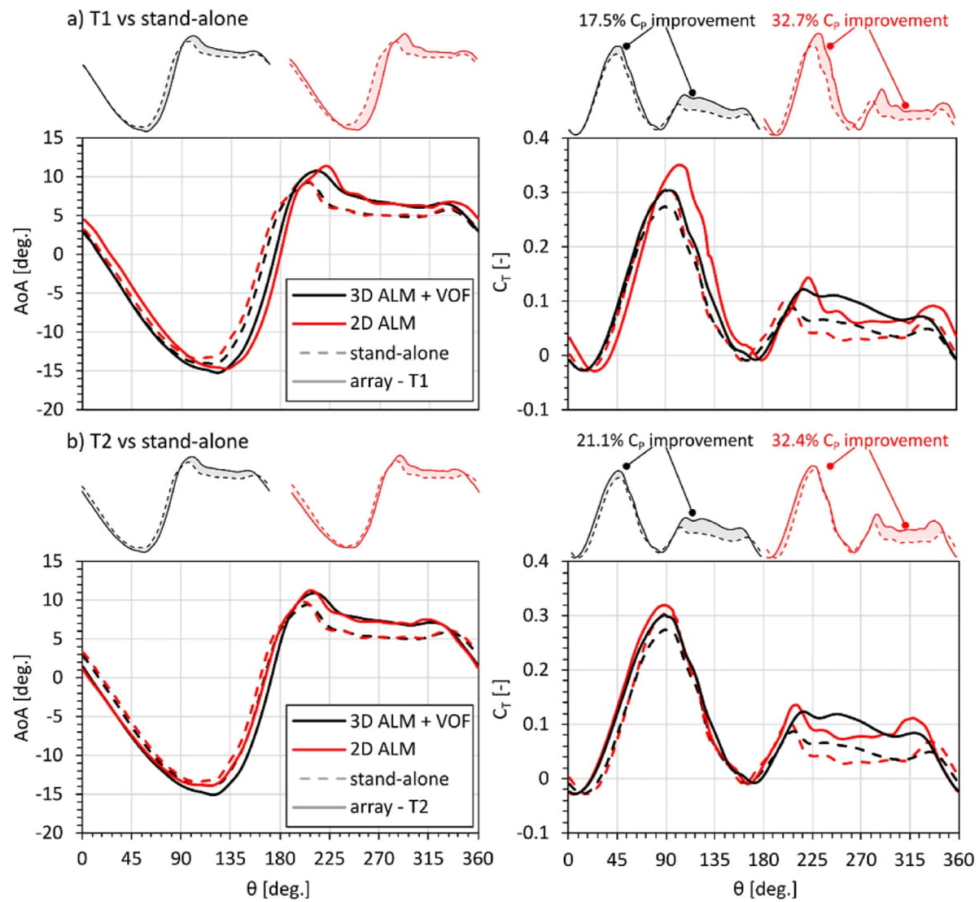


Fig. 24. Comparison between the 3D and the 2D ALM in terms of the capability of capturing the power augmentation in an array: stand-alone vs a) T1 and b) T2.

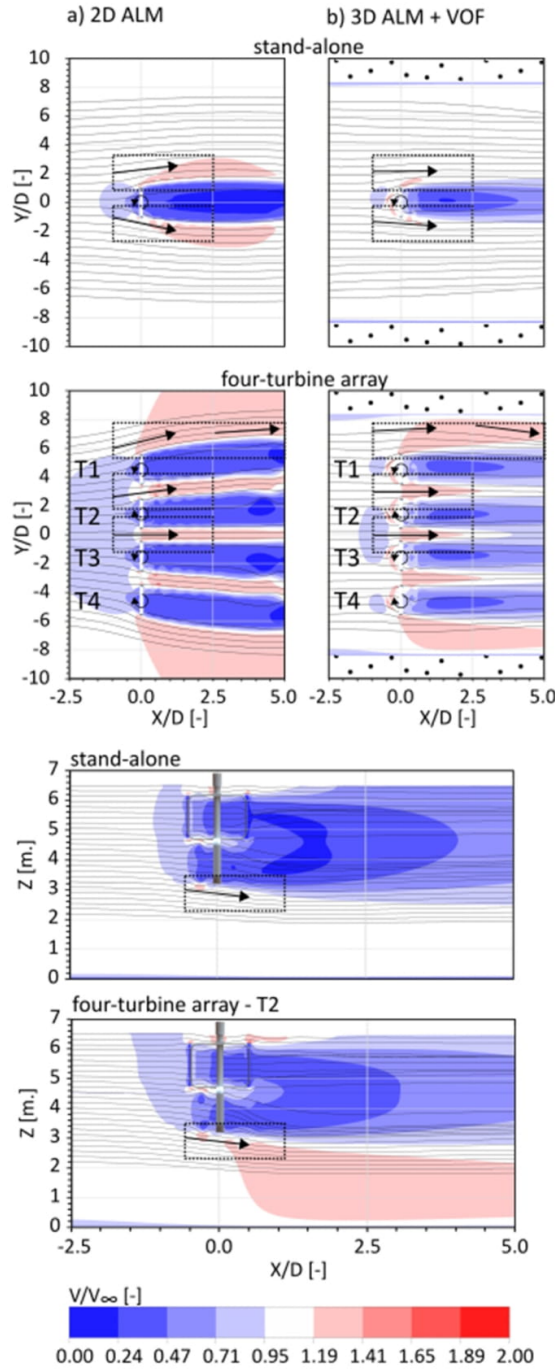


Fig. 25. Contours of dimensionless velocity and streamlines for the 2D ALM and the 3D ALM for stand-alone turbine and array.

wake area. In the case in hand, the acceleration zone did not present a significant risk to the channel, with $V_{max} \approx 1.2V_{\infty}$. However, this aspect must be carefully considered when designing an array of HKTs and tested for its impact, especially at high TSR operation of the turbines,

due to the potential long-term risks, such as erosion of the channel walls.

5.4. Power enhancement in arrays: 2D vs 3D

Fig. 23 presents the average cyclic C_p at different TSRs for the stand-alone turbine using three different simulation techniques: a) 2D blade-resolved CFD (Mohamed et al., 2023a), which is the conventional and widely used method for simulating Darrieus turbines; b) 2D ALM, using the latest formulation employed in the present and previous studies (Melani et al., 2021a; Mohamed et al., 2022, 2023b); and c) the present 3D ALM coupled with VOF.

Upon comparison of the results, one can notice that adding the three-dimensional effects as well as the influence of the free water surface makes the solution of the ALM close to that of the blade-resolved CFD. However, it can also be seen that at low TSRs, both the 2D and 3D ALM show a higher C_p , which is attributed to the known incapability of the Berg dynamic stall model to accurately reproduce the lift hysteresis, especially during the downwind half, resulting in overestimated power. This issue was previously discussed in detail by some of the authors in (Mohamed et al., 2022). The lower the TSR, the more this issue persists due to the relevance of the unsteady effects associated with the wide range of AoA seen by the turbine blades. At high TSRs, on the other hand, the presence of dynamic effects under attached flow conditions is also relevant. These effects are not considered in the Berg model, resulting in a slight overprediction of the power coefficient compared to blade-resolved CFD. (Melani et al., 2021a), (Mohamed et al., 2022), (Mohamed et al., 2023b)

As reported by some of the authors (Mohamed et al., 2023a) in a recent two-dimensional analysis using a blade-resolved CFD, deploying closely spaced Darrieus rotors in an inline layout creates a blockage that straightens the inflow and eliminates the streamtube expansion effect between adjacent rotors. This leads to an increase in the AoA and consequently, the lift force and produced torque, while also shifting the optimum power coefficient towards higher TSRs. Given that the present ALM code has demonstrated a strong capability in replicating this mechanism (Mohamed et al., 2023b), the power augmentation resulting from clustering the turbines can be more accurately assessed using the present CFD model.

To investigate this in detail, Fig. 24 illustrates the percentage improvement for each of the codes and compares their results for the instantaneous variations of the torque coefficient and angle of attack for a single blade of T1 and T2 in the four-turbine array against the stand-alone turbine. For T1, the power coefficient improvement is nearly doubled in the 2D and open (assumed infinite) domain due to the increased AoA within the mutual interaction area ($90^\circ > \theta > 270^\circ$). This increase in AoA, evidently is overestimated in the 2D case, especially in the second upwind quarter ($90^\circ > \theta > 180^\circ$). For T2, the differences between the 2D and 3D code estimations are less pronounced. It is important to note that for both T1 and T2, the performance improvement in the downwind half might be exaggerated, as detailed in (Mohamed et al., 2023b), which compared the present ALM code with blade-resolved CFD. It suggests that the discrepancy in blade loads during the downstream half is due to the ALM algorithm, particularly the Berg dynamic stall model, at which a good matching between the lift hysteresis was found between both ALM and blade-resolved simulations during pitch-up. However, in the downwind part, the ALM maintains a linear behaviour, whereas the computed blade-resolved data showed a reduction in the nominal slope and a lag in the instantaneous lift relative to the AoA.

Fig. 25 shows the dimensionless velocity contours and streamlines around both the stand-alone turbine and the four-turbine array, using 2D and 3D ALM. In the stand-alone turbine case, the 3D simulation does not exhibit the same degree of flow deviation in front of the turbine as the 2D simulation, where turbine blades are assumed to be infinite in the Z-direction. In the 2D case, this deviation is more apparent, showing more accelerated flow near the shear layers in the wake. Conversely, in

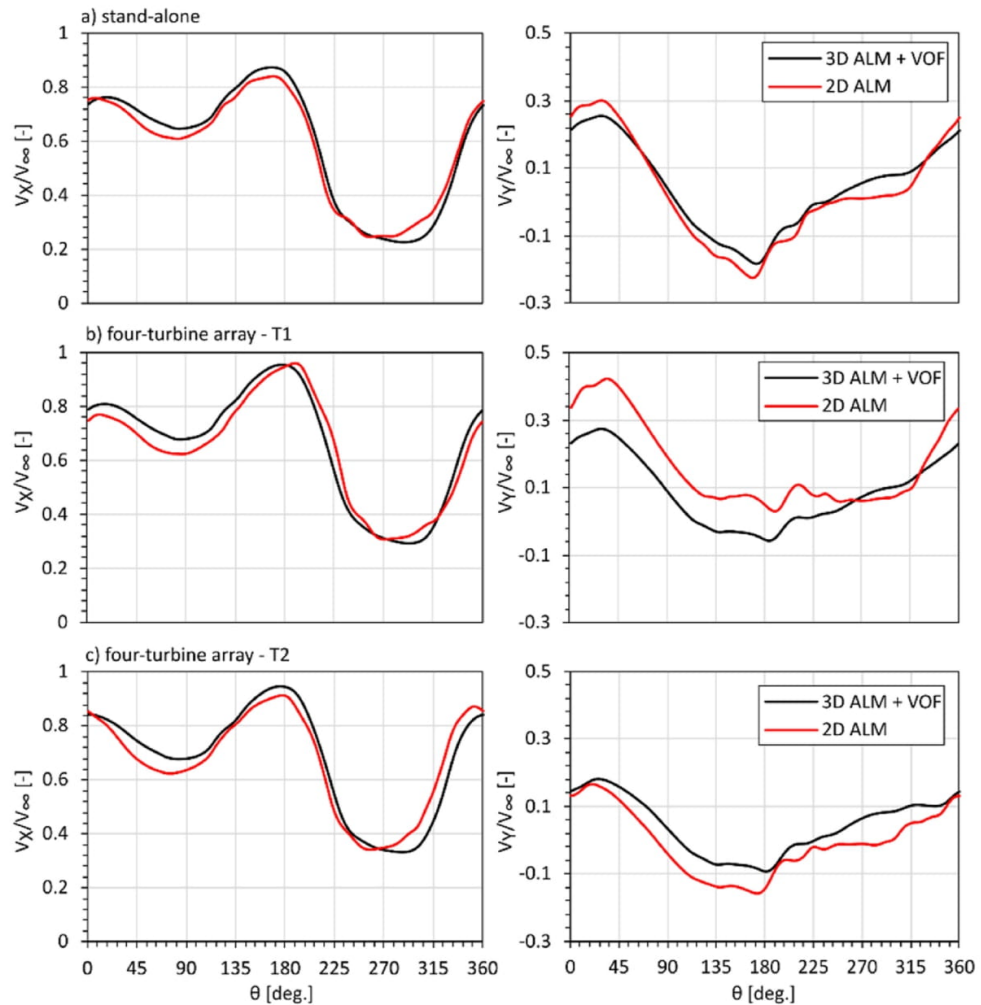


Fig. 26. Comparison between the V_x and V_y local to the blade in stand-alone and array 2D vs 3D ALM.

the 3D case, the elevation view in Fig. 19-b.2) reveals how the streamtube expansion is distributed among the three dimensions, with the flow deviating toward the ground. This reduces the impact of streamtube expansion in the plane view. This observation suggests that 2D simulations may be less valid for hydrokinetic turbines, given their relatively small aspect ratio compared to their wind counterparts. Additionally, the design of the channel itself plays an important role in this behaviour.

Moving to the four-turbine array, the behaviour of inflow deviation in front of the turbines on the sides differs significantly between the 3D and 2D cases. In the 3D scenario, the flow stream is confined by the channel walls, compelling it to straighten.

Conversely, in the 2D scenario, notable inflow deviation is observed on sides of the array, which accounts for the differences in sampled angle of attack variation for $T1$ between the 2D and 3D results. Additionally, at $X/D = 2.5$, the wake in the 3D case converges toward the wake core, while in the 2D case, it diverges. For $T2$, the differences in inflow deviation are less pronounced, with only a slightly more deviated inflow in the interaction area between $T1$ and $T2$, corresponding to the smaller differences in the estimated power coefficient of $T2$ between the 3D and 2D simulations. Notably, the degree of inflow vertical deviation

for $T2$ resembles that of the stand-alone turbine, despite the presence of a large acceleration zone extending up to the channel ground walls, imposed by the array blockage.

To elaborate on the previous observations, the local inflow velocity components are compared for the 2D and 3D ALM, these data were obtained using the LineAverage method explained in section 3.2.1. Fig. 26 shows the variation in the nondimensionalized X and Y velocities sampled in front of a single blade of $T1$ and $T2$ compared to the stand-alone turbine, obtained using both 2D and 3D ALM. For the stand-alone turbine, V_x is slightly higher in the 3D case, with lower V_y due to less flow deviation. For $T1$ in the array, a slight phase shift can be observed in the V_x of the 3D case, along with significant differences in V_y during the upwind half. This indicates less inflow deviation due to the blockage imposed by the channel walls in the first quarter ($0^\circ > \theta > 90^\circ$) and interaction with $T2$ in the second quarter ($90^\circ > \theta > 180^\circ$), explaining the significant differences in C_p and the shift in instantaneous C_T between the 2D and 3D results. The phase shift in the mutual interaction region between $T1$ and $T2$ is not captured by the 2D simulation because $T1$ is heavily affected by blockage due to channel walls. However, the V_x between $T2$ and $T3$ shows a similar trend in both 3D and 2D

simulations ($0^\circ > \theta > 90^\circ$ and $270^\circ > \theta > 360^\circ$ in the T2 results), suggesting minimal wall blockage effects. The primary blockage is caused by the turbine clustering, which is well captured in the 2D simulation.

6. Conclusions

This study introduces a medium-fidelity model for analysing the performance of an array of Darrieus-type HKTs in an open channel, employing the ALM coupled with the VOF method via U-RANS solver. A sensitivity analysis addressed the main modelling features for open channels and the necessary corrections for efficiency calculations considering water surface variations. Numerical sensitivity assessments were performed, followed by a validation campaign against experimental measurements for turbine performance, water level, and flow field velocities. Then, the study delved into turbine performance and the power augmentation mechanism from clustering turbines in close proximity. Furthermore, the hydraulic impact of the turbine array on the channel was examined, addressing potential concerns when installing turbine arrays in channels.

Overall, among the various simulation techniques for the Darrieus turbine, the proposed CFD model offers the best trade-off between accuracy and computational time. The following points summarize the main findings of the study:

- Coupling the ALM with the VOF can improve predictions of turbine performance, particularly at high TSRs, by accounting for the variations in the free water surface. However, at low TSRs, the accuracy of the ALM decreases due to unsteady effects that are not fully captured by the Berg dynamic stall model used for the present analysis.
- Good agreement was found between the experimental velocity measurements and the ALM results, especially in the near wake area. However, discrepancies in the longitudinal wake measurements at the turbine tip were observed, possibly due to lack of struts modelling. This issue will be addressed in future work.
- When simulating HKTs assuming a horizontal-bed open channel, the initial inlet velocity should be used to compute turbine power coefficients, as velocities in front of turbines are affected by blockage, e.g., $\approx 0.9V_{\text{inlet}}$ for single turbine and $\approx 0.86V_{\text{inlet}}$ for the four-turbine cluster, thus overestimating efficiency.
- When doing a CFD simulation for HKTs in open channels, greater distance between turbines and inlet is necessary for flow to fully develop, while distance between turbines and outlet is less critical.
- The HKT array in this study resulted in up to 25% more power than stand-alone turbines due to combined blockage effects from channel walls and neighbouring turbines. Additionally, the optimum TSR shifted toward a higher value.
- Realistic simulations considering channel boundaries and free water surface show significant differences between 2D and 3D turbine models, with 2D simulations exaggerating power augmentation effects, e.g., 32% maximum power augmentation in 2D compared to a corresponding 21% for the 3D.
- Although installing Darrieus-type HKTs in close proximity enhances efficiency, turbine arrays create acceleration zones near channel walls more than stand-alone turbines, requiring careful consideration for long-term impacts on the channel. The use of 3D models like the one proposed here is strongly recommended in these applications.

Future work should focus on enhancing the accuracy of ALM predictions, particularly at low TSRs, by incorporating more advanced dynamic stall models. Additionally, a more thorough investigation of tip effects in simulations is needed to improve overall model fidelity.

CRediT authorship contribution statement

Omar S. Mohamed: Writing – original draft, Visualization, Validation, Software, Methodology, Investigation, Formal analysis, Data curation. **Pier Francesco Melani:** Writing – review & editing, Visualization, Software, Methodology. **Giuseppe Soraperra:** Writing – review & editing, Resources, Investigation. **Alessandra Brighenti:** Writing – review & editing, Resources, Investigation. **Giovanni Ferrara:** Writing – review & editing, Supervision, Project administration. **Vittorino Betti:** Project administration, Funding acquisition. **Leonardo Schippa:** Writing – review & editing, Investigation. **Massimo Guerrero:** Writing – review & editing, Investigation. **Francesco Balduzzi:** Writing – review & editing, Methodology, Conceptualization. **Alessandro Bianchini:** Writing – review & editing, Resources, Project administration, Methodology, Investigation, Funding acquisition, Data curation, Conceptualization.

Declaration of competing interest

The authors declare that they have no known competing financial interests or personal relationships that could have appeared to influence the work reported in this paper.

Acknowledgments

Thanks are due to HE-Powergreen for the support in the PhD of Omar Sherif Mohamed and for sharing experimental measurements.

References

- Bachant, P., Goude, A., Wosnik, M., 2018. Actuator Line Modeling of Vertical-axis Turbines. <https://doi.org/10.48550/arXiv.1605.01449> arXiv, arXiv:1605.01449.
- Balduzzi, F., Bianchini, A., Maleci, R., Ferrara, G., Ferrari, L., 2016a. Critical issues in the CFD simulation of Darrieus wind turbines. *Renew. Energy* 85, 419–435. <https://doi.org/10.1016/j.renene.2015.06.048>.
- Balduzzi, F., Bianchini, A., Ferrara, G., Ferrari, L., 2016b. Dimensionless numbers for the assessment of mesh and timestep requirements in CFD simulations of Darrieus wind turbines. *Energy* 97, 246–261. <https://doi.org/10.1016/j.energy.2015.12.111>.
- Balduzzi, F., Melani, P.F., Soraperra, G., Brighenti, A., Battisti, L., Bianchini, A., 2021. Some design guidelines to adapt a Darrieus vertical axis turbine for use in hydrokinetic applications. *E3S Web Conf.* 312, 08017. <https://doi.org/10.1051/e3sconf/202131208017>.
- Cacciali, L., Battisti, L., Dell'Anna, S., 2022. Free surface double actuator disc theory and double Multiple streamtube model for in-stream Darrieus hydrokinetic turbines. *Ocean Eng.* 260, 112017. <https://doi.org/10.1016/j.oceaneng.2022.112017>.
- Cacciali, L., Battisti, L., Dell'Anna, S., 2023a. Backwater assessment for the energy conversion through hydrokinetic turbines in subcritical prismatic canals. *Ocean Eng.* 267, 113246. <https://doi.org/10.1016/j.oceaneng.2022.113246>.
- Cacciali, L., Battisti, L., Dell'Anna, S., 2023b. Multi-array design for hydrokinetic turbines in hydropower canals. *Energies* 16 (5), 5. <https://doi.org/10.3390/en16052279>.
- Chaulagain, R.K., Poudel, L., Maharjan, S., 2023. A review on non-conventional hydropower turbines and their selection for ultra-low-head applications. *Heliyon* 9 (7), e17753. <https://doi.org/10.1016/j.heliyon.2023.e17753>.
- Chen, Y., Guo, P., Zhang, D., Chai, K., Zhao, C., Li, J., 2022. Power improvement of a cluster of three Savonius wind turbines using the variable-speed control method. *Renew. Energy* 193, 832–842. <https://doi.org/10.1016/j.renene.2022.05.062>.
- de Boor, C., 1978. A practical guide to splines. In: *Applied Mathematical Sciences*. Springer-Verlag, New York [Online]. Available: <https://www.springer.com/gp/book/9780387953663>. (Accessed 23 November 2020).
- Draper, S., Nishino, T., 2014. Centred and staggered arrangements of tidal turbines. *J. Fluid Mech.* 739, 72–93. <https://doi.org/10.1017/jfm.2013.593>.
- Gauvin-Tremblay, O., Dumas, G., 2020. Two-way interaction between river and deployed cross-flow hydrokinetic turbines. *J. Renew. Sustain. Energy* 12 (3), 034501. <https://doi.org/10.1063/5.0004492>.
- Guerrero, M., Conevski, S., Cavalieri, I., Schippa, L., Ruther, N., 2023. Influence of hydropower propellers on open-channel flow. *J. Hydraul. Eng.* 149 (11), 04023047. <https://doi.org/10.1061/JHEND8.HYENG-13417>.
- Hirt, C.W., Nichols, B.D., 1981. Volume of fluid (VOF) method for the dynamics of free boundaries. *J. Comput. Phys.* 39 (1), 201–225. [https://doi.org/10.1016/0021-9991\(81\)90145-5](https://doi.org/10.1016/0021-9991(81)90145-5).
- Jost, E., Klein, L., Leipprand, H., Lutz, T., Krämer, E., 2018. Extracting the angle of attack on rotor blades from CFD simulations. *Wind Energy* 21 (10), 807–822. <https://doi.org/10.1002/we.2196>.
- Kirke, B., 2019. Hydrokinetic and ultra-low head turbines in rivers: a reality check. *Energy Sustain. Deve.* 52, 1–10. <https://doi.org/10.1016/j.esd.2019.06.002>.

- Marten, D., et al., 2016. Effects of airfoil's polar data in the stall region on the estimation of Darrieus wind turbines performance. *J. Eng. Gas Turbines Power* 139 (2), 22606–22609. <https://doi.org/10.1115/1.4034326>.
- Massé, B., 1981. Description de deux programmes d'ordinateur pour le calcul des performances et des charges aérodynamiques pour les éoliennes à axe vertical technical report IREQ-2379.
- Melani, P.F., 2018. Experimental assessment of an actuator line simulation tool for vertical axis wind turbines [Online]. Available: <https://www.politesi.polimi.it/handle/10589/141629>. (Accessed 5 September 2024).
- Melani, P.F., Balduzzi, F., Ferrara, G., Bianchini, A., 2020. How to extract the angle attack on airfoils in cycloidal motion from a flow field solved with computational fluid dynamics? Development and verification of a robust computational procedure. *Energy Convers. Manag.* 223, 113284. <https://doi.org/10.1016/j.enconman.2020.113284>.
- Melani, P.F., Balduzzi, F., Ferrara, G., Bianchini, A., 2021a. Tailoring the actuator line theory to the simulation of Vertical-Axis Wind Turbines. *Energy Convers. Manag.* 243, 114422. <https://doi.org/10.1016/j.enconman.2021.114422>.
- Melani, P.F., Balduzzi, F., Ferrara, G., Bianchini, A., 2021b. Development of a desmodromic variable pitch system for hydrokinetic turbines. *Energy Convers. Manag.* 250, 114890. <https://doi.org/10.1016/j.enconman.2021.114890>.
- Melani, P.F., Balduzzi, F., Bianchini, A., 2021c. A robust procedure to implement dynamic stall models into actuator line methods for the simulation of vertical-axis wind turbines. *J. Eng. Gas Turbines Power* 143 (11). <https://doi.org/10.1115/1.4051909>.
- Melani, P.F., Balduzzi, F., Bianchini, A., 2022. Simulating tip effects in vertical-axis wind turbines with the actuator line method. *J. Phys.: Conf. Ser.* 2265 (3), 032028. <https://doi.org/10.1088/1742-6596/2265/3/032028>.
- Melani, P.F., Mohamed, O.S., Cioni, S., Balduzzi, F., Bianchini, A., 2024a. An insight into the capability of the actuator line method to resolve tip vortices. *Wind Energy Sci.* 9 (3), 601–622. <https://doi.org/10.5194/wes-9-601-2024>.
- Melani, P.F., Mohamed, O.S., Cioni, S., Balduzzi, F., Bianchini, A., 2024b. Tuning the actuator line method to properly modelling tip effects in finite-length blades. *J. Phys.: Conf. Ser.* 2767 (5), 052018. <https://doi.org/10.1088/1742-6596/2767/5/052018>.
- Menter, F.R., 1993. Zonal two equation kappa-omega turbulence models for aerodynamic flows. Presented at the AIAA Fluid Dynamics Conference. Orlando, FL. <https://ntrs.nasa.gov/citations/19960044572>. (Accessed 19 April 2023).
- Menter, F.R., Smirnov, P.E., Liu, T., Avancha, R., 2015. A one-equation local correlation-based transition model. *Flow, Turbul. Combust.* 95 (4), 583–619. <https://doi.org/10.1007/s10494-015-9622-4>.
- Mohamed, O.S., Ibrahim, A., El Baz, A.M.R., 2019. CFD investigation of the Multiple rotors Darrieus type turbine performance. Presented at the ASME Turbo Expo 2019: Turbomachinery Technical Conference and Exposition. American Society of Mechanical Engineers Digital Collection. <https://doi.org/10.1115/GT2019-91491>.
- Mohamed, O.S., Elbaz, A.M.R., Bianchini, A., 2021. A better insight on physics involved in the self-starting of a straight-blade Darrieus wind turbine by means of two-dimensional Computational Fluid Dynamics. *J. Wind Eng. Ind. Aerod.* 218, 104793. <https://doi.org/10.1016/j.jweia.2021.104793>.
- Mohamed, O.S., Melani, P.F., Balduzzi, F., Ferrara, G., Bianchini, A., 2022. An insight on the key factors influencing the accuracy of the actuator line method for use in vertical-axis turbines: limitations and open challenges. *Energy Convers. Manag.* 270, 116249. <https://doi.org/10.1016/j.enconman.2022.116249>.
- Mohamed, O.S., Melani, P.F., Balduzzi, F., Ferrara, G., Bianchini, A., 2023a. An insight on the physical mechanisms responsible of power augmentation in a pair of counter-rotating Darrieus turbines. *Energy Convers. Manag.* 284, 116991. <https://doi.org/10.1016/j.enconman.2023.116991>.
- Mohamed, O.S., Melani, P.F., Papi, F., Balduzzi, F., Bianchini, A., 2023b. Is the Actuator line method able to reproduce the interaction between closely-spaced Darrieus rotors? a critical assessment on wind and hydrokinetic turbines. *Energy Convers. Manag.* 293. <https://doi.org/10.1016/j.enconman.2023.117473>.
- Myers, L., Bahaj, A.S., 2007. Wake studies of a 1/30th scale horizontal axis marine current turbine. *Ocean Eng.* 34 (5), 758–762. <https://doi.org/10.1016/j.oceaneng.2006.04.013>.
- Nag, A.K., Sarkar, S., 2021. Performance analysis of Helical Savonius Hydrokinetic turbines arranged in array. *Ocean Eng.* 241, 110020. <https://doi.org/10.1016/j.oceaneng.2021.110020>.
- Niebuhr, C.M., van Dijk, M., Neary, V.S., Bhagwan, J.N., 2019. A review of hydrokinetic turbines and enhancement techniques for canal installations: technology, applicability and potential. *Renew. Sustain. Energy Rev.* 113, 109240. <https://doi.org/10.1016/j.rser.2019.06.047>. Oct.
- Nishi, Y., Sato, G., Shiohara, D., Inagaki, T., Kikuchi, N., 2017. Performance characteristics of axial flow hydraulic turbine with a collection device in free surface flow field. *Renew. Energy* 112, 53–62. <https://doi.org/10.1016/j.renene.2017.04.047>.
- Nishino, T., Willden, R.H.J., 2012. The efficiency of an array of tidal turbines partially blocking a wide channel. *J. Fluid Mech.* 708, 596–606. <https://doi.org/10.1017/jfm.2012.349>.
- Nishino, T., Willden, R.H.J., 2013. Two-scale dynamics of flow past a partial cross-stream array of tidal turbines. *J. Fluid Mech.* 730, 220–244. <https://doi.org/10.1017/jfm.2013.340>.
- Rainbird, J.M., et al., 2015. On the influence of virtual camber effect on airfoil polars for use in simulations of Darrieus wind turbines. *Energy Convers. Manag.* 106, 373–384. <https://doi.org/10.1016/j.enconman.2015.09.053>.
- Sørensen, J.N., Shen, W.Z., 2002. Numerical modeling of wind turbine wakes. *J. Fluid Eng.* 124 (2), 393–399. <https://doi.org/10.1115/1.1471361>.
- Sørensen, J.N., Dag, K.O., Ramos-García, N., 2014. A new tip correction based on the decambering approach. *J. Phys.: Conf. Ser.* 524 (1), 012097. <https://doi.org/10.1088/1742-6596/524/1/012097>.
- Viterna, L.A., Janetzke, D.C., 1982. Theoretical and Experimental Power from Large Horizontal-Axis Wind Turbines. NASA Technical Memorandum.
- Yadav, P.K., Kumar, A., Jaiswal, S., 2023. A critical review of technologies for harnessing the power from flowing water using a hydrokinetic turbine to fulfill the energy need. *Energy Rep.* 9, 2102–2117. <https://doi.org/10.1016/j.egy.2023.01.033>.
- Yosry, A.G., Álvarez, E.A., Valdés, R.E., Pandal, A., Marigorta, E.B., 2023. Experimental and multiphase modeling of small vertical-axis hydrokinetic turbine with free-surface variations. *Renew. Energy* 203, 788–801. <https://doi.org/10.1016/j.renene.2022.12.114>.
- Zanforlin, S., Nishino, T., 2016. Fluid dynamic mechanisms of enhanced power generation by closely spaced vertical axis wind turbines. *Renew. Energy* 99, 1213–1226. <https://doi.org/10.1016/j.renene.2016.08.015>.
- ANSYS FLUENT 12.0 Theory Guide'. Accessed: November, 20, 2023. [Online]. Available: https://www.afs.enea.it/project/neptunius/docs/fluent/html/th/main_pre.htm.
- Il Canale Biffis - Claudio Malini - Libro - Sometti - Acque e bonifiche | IBS. <https://www.ibs.it/canale-biffis-libro-claudio-malini/e/9788874956760>. (Accessed 22 February 2024).



Contents lists available at ScienceDirect

Ocean Engineering

journal homepage: www.elsevier.com/locate/oceaneng

Research paper

Development of a computationally efficient CFD model for vertical-axis hydrokinetic turbines: A critical assessment

Omar S. Mohamed¹ , Pier Francesco Melani¹ , Giuseppe Soraperra²,
Alessandra Brighenti², Giovanni Ferrara¹, Vittorino Betti², Francesco Balduzzi¹ ,
Alessandro Bianchini^{1,*}

¹ Department of Industrial Engineering, Università degli Studi di Firenze, Via di Santa Marta 3, 50139, Firenze, Italy

² HE-PowerGreen s.r.l., Via Solteri 74, 38121, Trento, Italy

ARTICLE INFO

Keywords:

Actuator Cylinder (AC)
Darrieus
Vertical Axis Turbine (VAT)
Hydrokinetic turbines
Computational Fluid Dynamics (CFD)
Volume of Fluid (VOF)

ABSTRACT

Darrieus-type vertical axis turbines are suitable for hydrokinetic applications, particularly when deployed in arrays within rivers and small watercourses. However, modelling their complex and unsteady hydrodynamic interactions, both among turbines and with channel boundaries and free water surface, remains a significant challenge. This study develops a computationally-efficient Actuator Cylinder (AC) model that is coupled for the first time with the Volume of Fluid (VOF) method to account for free water surface variations. The proposed AC model calculates instantaneous blade forces, but simplifies them by projecting only the average forces, eliminating dependence on time-step size. The new approach, developed with the CFD code Ansys Fluent, has been validated against experiments and benchmarked with a higher-fidelity Actuator Line Method (ALM). Results indicate that the AC model provides reasonably accurate estimates of turbine performance, as well as the impact of the turbines on the flow field within the channel. For a stand-alone turbine, the AC model shows close alignment with the ALM, with a maximum difference of only 3.5% for the power coefficient. However, in array simulations, the AC model tends to underestimate the performance. Compared to the ALM, which more closely matches experimental data, the AC model overestimates the upstream water rise due to turbine-channel interactions and underestimates the power augmentation seen in Darrieus turbine arrays. Overall, the AC model offers an acceptable balance between accuracy and computational efficiency, significantly reducing computational time by one order of magnitude compared to the ALM. This makes the proposed CFD-AC-VOF model a practical tool for rapid and cost-effective analysis, in particular for the industrial sector.

1. Introduction

Traditional hydroelectric power generation relies on the conversion of the potential energy of descending water, with the potential capacity governed by flow rate and head. This is known as hydrostatic energy conversion. Despite its superior efficiency compared to other renewable energy resources, this method requires specific geographic features and faces economic and environmental limitations (Yadav et al., 2023). Recently, attention has been shifted to hydrokinetic turbines (HKTs), which capture the kinetic energy of water currents with minimal head loss, similar to wind energy (Kirke, 2019). HKTs can operate in diverse water environments, including tidal zones, ocean currents, waves, rivers, and small watercourses (Yuce and Muratoglu, 2015). Although

their power output is lower than conventional hydroelectric turbines, HKTs offer advantages for distributed and remote energy systems, they are easier to control, and they have minimal environmental impact. Moreover, they can even be installed downstream of existing hydroelectric plants (Holanda et al., 2017; Saini et al., 2021).

Similar to wind turbines, HKTs are classified based on the orientation of their axis of rotation relative to the water flow, i.e., horizontal-axis turbines (HATs) and vertical-axis turbines (VATs). While VATs are typically less efficient than HATs (Niebuhr et al., 2019), they offer easier maintenance due to the generator being located outside the water. Additionally, deploying multiple VATs in close proximity can increase the overall efficiency of the array due to turbine-to-turbine interactions (Mohamed et al., 2023a, 2024a). However, the potential power gains

* Corresponding author.

E-mail address: alessandro.bianchini@unifi.it (A. Bianchini).

<https://doi.org/10.1016/j.oceaneng.2025.120692>

Received 21 December 2024; Received in revised form 1 February 2025; Accepted 13 February 2025

0029-8018/© 2025 The Authors. Published by Elsevier Ltd. This is an open access article under the CC BY license (<http://creativecommons.org/licenses/by/4.0/>).

from turbine arrays are constrained by interactions between the turbines and channel hydraulics.

Studies by (Yosry et al., 2023; Gauvin-Tremblay and Dumas, 2020) have examined stand-alone Darrieus-type HKTs using blade-resolved Computational Fluid Dynamics (CFD) coupled with the Volume of Fluid (VOF) method to simulate their effects on the free water surface. These studies observed that when turbine-to-channel interactions are considered, power extraction becomes nearly independent from the turbine's immersion level. They emphasised the importance of accounting for the upstream water rise and its impact on the flow onset velocity, which is critical when computing turbine efficiency. In particular (Yosry et al., 2023), found that, for a blockage ratio between 15% and 23.8%, the power coefficient obtained from the CFD model that considers the free water surface was 90%–92% lower than that from a single-phase model, depending on the tip-speed ratio. Similarly (Gauvin-Tremblay and Dumas, 2020), observed that for a blockage ratio of 20%, the turbine efficiency, when accounting for losses due to free water surface effects, should be corrected to be 93%–96% of the resulted power, depending on the immersion level relative to the water surface. In a study by the authors (Mohamed et al., 2024b), the VOF method was combined with the Actuator Line Method (ALM) to model turbine forces. For a stand-alone turbine with an 8% blockage ratio, the power coefficient correction was consistently low, around 99%. However, for a four-turbine array with a blockage ratio of 31%, the power coefficient correction ranged between 96% and 97%, depending on the tip-speed ratio.

Thus, unlike wind turbines, HKTs in arrays must be simulated together with their corresponding channels, as turbine-to-turbine and turbine-to-channel interactions are crucial for accurately assessing array performance. Ignoring these effects when calculating the power coefficient can lead to overestimations due to blockage caused by confined channel boundaries. Since blade-resolved CFD is impractical for studying turbine arrays under such complex realistic conditions, low-cost computational approaches are required. Among them, the so called “hybrid models” are particularly promising. These models avoid explicitly resolving the turbine blade boundary layer, significantly reducing computational time while enabling the simulation of channel hydraulics and capturing the two-way interaction between the HKT array and the channel.

In this perspective, the authors recently developed a coupled CFD-VOF approach combined with the ALM to simulate an array of Darrieus HKTs in open-channel conditions (Mohamed et al., 2024b). This model was successfully validated against experimental data regarding turbine performance, upstream and downstream velocity deficits, and upstream water rise. Despite its lower computational cost compared to blade-resolved CFD, it remains computationally intensive, especially for industrial needs, where a fast computational tool is required for a rapid performance estimation, channel hydraulic analysis, and layout optimisation.

Several aero/hydrodynamic models with a lower cost than the ALM have been developed for VATs, each with trade-offs between accuracy and computational efficiency due to the complex flow patterns around the turbines. Many of these models are built on principles from the well-established actuator disk theory used in horizontal axis wind turbines. For instance, the Double Multiple Streamtube (DMST) model, introduced by (Paraschivoiu, 2002), approximates VAT dynamics by dividing the flow into multiple streamtubes. On the other hand, the Actuator Cylinder (AC) model, introduced by (Madsen, 1982), extends the actuator disk concept by representing the VAT rotor as a continuous cylinder, enabling better modelling of wake interactions and blockage effects.

In a DMST-based model (Cacciali et al., 2022, 2023a), introduced an iterative algorithm to adjust the upstream inflow of a Darrieus hydrokinetic turbine based on downstream conditions in a free-surface channel. The AC concept was further developed by (Madsen et al., 2014) in combination with blade element analysis, and by (Madsen et al., 2014) to simulate the performance of a floating Darrieus-type.

VAT (Ning, 2016) expanded the AC model to account for multiple turbines, while (Shives et al., 2017) focused on wake interactions and channel blockage effects in HKT arrays, though their approach neglected the influence of the free water surface. Although the DMST is relatively fast compared to other method, it suffers from reliability issues. For instance, in a dedicated comparison (Ferreira et al., 2014), evaluated DMST- and AC-based models, among others, and demonstrated that the DMST-based models are inconsistent across various operating and geometric conditions. Specifically, their sensitivity to different solidities and blade pitching makes them less reliable than AC-based models.

In the present work, a quasi-steady approach based on the AC concept was developed to simulate the performance of an array of Darrieus HKTs in a free-surface channel, aiming at reducing the computational time. Unlike any other approach in the literature to date, the AC model is coupled with the VOF method to capture the free water surface dynamics and simulate the interaction between the turbines and the channel. This follows the successful coupling of the ALM with VOF in a recent work by the authors (Mohamed et al., 2024b). The AC model represents the turbines as cylinders using an element-weighted approach, distributing the average forces across each AC element based on variations in instantaneous blade forces. Given the fundamental differences between the AC and ALM models in how turbines are represented spatially and temporally, the comparison between the AC and ALM models, both coupled with VOF in a CFD framework, and the subsequent validation against experimental measurements represents an element of novelty and scientific interest.

A sensitivity analysis was conducted to optimize the AC model settings, including the angle of attack sampling radius, the size of the regularization kernel for force projection, and the number of elements constructing the actuator cylinder. The model was validated against experimental measurements (Guerrero et al., 2023), including data on free water surface levels, turbine performance, and flow velocities upstream and downstream of the turbine array. Additionally, a detailed comparison was performed to highlight the differences in turbine-to-turbine interactions between the AC and the ALM recently presented by (Mohamed et al., 2024b). Finally, an evaluation of the hydraulic impact of the turbines on the channel flow was carried out for both stand-alone and array configurations.

The paper is structured as follows. Section 2 describes the geometry and operational details of the four-turbine array, along with the hydraulic characteristics of the channel. Section 3 presents the AC model, including the sensitivity analysis on its settings, and details the CFD setup. Section 4 presents the results and discussion, featuring a validation against experimental data and a detailed comparison with the ALM model. Finally, Section 5 concludes the paper and proposes future research directions.

2. Test case

2.1. Hydrokinetic turbine model

The turbine model featured in this study (whose main characteristics are reported in Table 1) is a two-bladed H-Darrieus HKT, consisting of two connected rotors with 2 m diameter and a total height of 3 m, as illustrated in Fig. 1-a. This design was developed in collaboration with the industrial partner HE-Powergreen as part of previous research

Table 1
Geometrical characteristics of the test case turbine.

Dimension	Value
Number of blades [-]	2
Diameter, D [m]	2
Turbine height, H [m]	3
Aspect ratio (H/D)	1.5
Solidity (Nc/D)	0.25

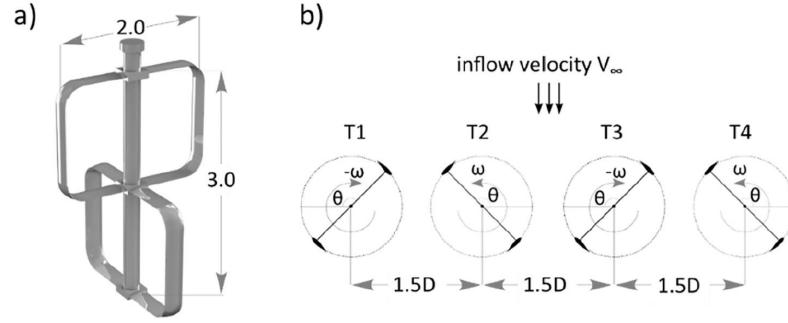


Fig. 1. a) schematic representation of the turbine model and b) the 4-turbine array used in the study.

(Balduzzi et al., 2021). The turbine is segmented at the midpoint, allowing the upper and lower rotors to be oriented with a 90° azimuthal offset. This configuration enhances mechanical stability and minimizes torque ripples. The blades incorporate a proprietary hydrofoil based on the DU-06-W-200 profile, modified for thickness distribution and adjusted for flow curvature effects (Rainbird et al., 2015). Similar turbine configurations have been simulated with an in-house ALM code in past studies (Melani et al., 2021b; Mohamed et al., 2022, 2023b), demonstrating strong agreement with blade-resolved CFD.

This turbine model is simulated herein in a stand-alone and four-turbine array configurations. Fig. 1(b) shows the rotational arrangements adopted for the array, with each turbine counter-rotating against its neighbouring turbine. A consistent centre-to-centre spacing of 1.5 times the rotor diameter is maintained between turbines. The operational parameters of the turbines are listed in Table 2.

2.2. Channel description

Fig. 2 illustrates the channel used in this study, which is a section of the Biffis artificial channel near Trento, Italy (Il Canale Biffis - Claudio Malini - Libro - Sometti - Acque e bonifiche | IBS, 2024). The channel has a double trapezoidal cross-section, approximately 21.65 m wide, with a wetted width of about 20.2 m and an average flow rate of $124 \text{ m}^3/\text{s}$. The bottom width is around 4.45 m, and the maximum height from the bottom to the top is 7.2 m, with a wetted height of 6.52 m. Additional details about the channel can be found in previous work by (Cacciali et al., 2023b). The real channel maintains this shape, with slight variations in dimensions upstream and downstream of the section where the turbine array is installed.

3. Methodology

3.1. Actuator cylinder algorithm

The proposed Actuator Cylinder (AC) algorithm aims to convert the time-dependent, in-house ALM code from (Melani et al., 2021c) into a simplified, steady model optimised for the rapid calculation of turbine performance and the surrounding flow field. The AC algorithm generally follows the structure of the ALM approach, wherein both methods do not explicitly resolve the flow around the blade geometry. Instead, they

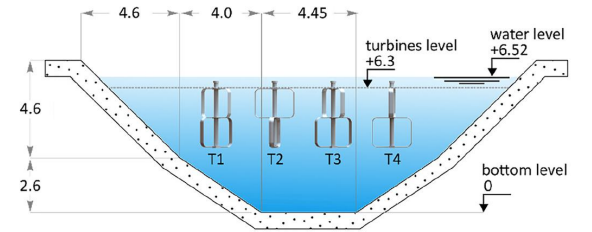


Fig. 2. Biffis channel cross section and dimensions in meters.

compute and insert turbine forces into the CFD domain as equivalent momentum sources based on local kinematic data sampled from the resolved CFD flow field. However, in the AC algorithm, blade forces are averaged over a complete revolution before being projected onto the computational domain, resulting in a solution independent from time step size.

In this model, as illustrated in Fig. 3, the turbine is represented as a cylinder divided into n sampling points and m actuator elements. It is important to note that sampling points and actuator elements are discretized independently, using angular step sizes $\Delta\theta$ and $\Delta\phi$, respectively, where θ is the position of the sampling point and ϕ is the position of the actuator element centre. This distinction arises because a high resolution is required for the angle of attack data, whereas forces can be projected with a coarser resolution, as detailed in Section 4.3. The number of elements in the spanwise direction z is based on the mesh sizing, which is not uniform but decreases gradually in element size toward the tip.

First, the flow field, characterized by velocity components in the X and Y directions, is sampled across each n point, and the normal W_n and tangential W_t velocity components are computed as in Eqs. (1) and (2). This sampling process effectively captures the spatial variations in flow kinematics. Next, the azimuthal variations of the angle of attack $\alpha(\theta)$ and chord-based Reynolds number Re_c are calculated using Eqs. (3) and (4):

$$W_n = -V_x \sin(\theta) + V_y \cos(\theta) \quad (1)$$

$$W_t = -V_x \cos(\theta) - V_y \sin(\theta) - \omega R \quad (2)$$

$$\alpha = \tan^{-1} \left(-\frac{W_n}{W_t} \right) \quad (3)$$

$$Re_c = \frac{\rho W_c}{\mu} \quad (4)$$

This information is used to retrieve the hydrofoil polar data, which is provided as pre-prepared tables of lift C_L , drag C_D , and pitching moment C_M coefficients.

Table 2
Test operating conditions.

Parameter	Value
Tip-speed ratio range [-]	2.23–3.26
Working fluid [-]	water
Inflow speed, V_∞ [m/s]	1.6
Inflow turbulence intensity [%]	0.24
Chord-based Reynolds number range, Re_c [-]	$\approx 420k - 1,600k$

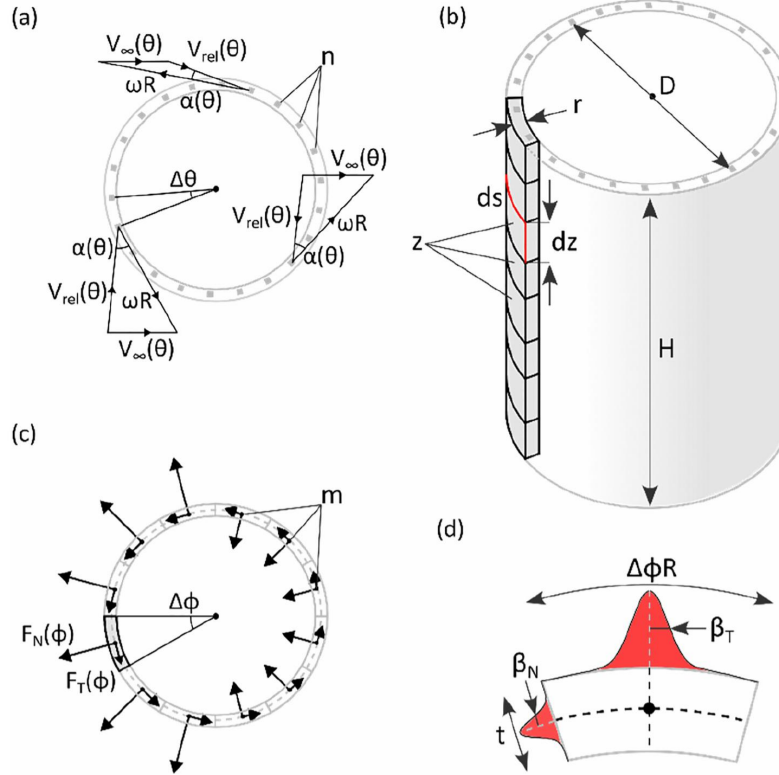


Fig. 3. Schematic representation of the Actuator Cylinder (AC).

The quality of polar data plays a crucial role in the accuracy of ALM predictions, particularly for small Darrieus turbines. Prior studies (Melani et al., 2021a) have highlighted the significance of laminar separation bubbles under conditions where the chord-based Reynolds number is low. In the present study, polar data were obtained up to the stall limit using 2D blade-resolved CFD with the $k-\omega$ SST turbulence model. An additional equation for intermittency transport was used to account for laminar-to-turbulent transition, obtaining the so-called γ model (Menter et al., 2015). For deep stall conditions (up to $\pm 180^\circ$ angle of attack), the Viterna-Corrigan extrapolation (Viterna and Janetzke, 1982) was employed, following the recommendations of (Marten et al., 2016). To account for unsteady effects on the blade load, the sampled kinematic data were treated as transient-like, as their variation along the azimuthal spatial path is analogous to temporal variation.

The time step size Δt , used later to compute the unsteady loads, is calculated based on the resolution of the n points (corresponds to $\Delta\theta = 1^\circ$) and the turbine rotational speed ω , as in Eq (5).

$$\Delta t = \frac{2\pi}{\Delta\theta\omega} \quad (5)$$

The equivalent polar data at each n point is then used to compute the corresponding normal $F_N(\theta)$ and tangential $F_T(\theta)$ forces. To compute the cyclic mean performance, the average turbine forces are calculated as shown in Eqs (6) and (7):

$$\bar{F}_N = \frac{B}{n} \sum_{i=1}^n \sum_{k=1}^z F_{N,ik} \Delta\theta R dz_k \quad (6)$$

$$\bar{F}_T = \frac{B}{n} \sum_{i=1}^n \sum_{k=1}^z F_{T,ik} \Delta\theta R dz_k \quad (7)$$

where B is the number of blades, and the arc $\Delta\theta R$ represents the spatial azimuthal step size based on the resolution of the sampled kinematics. To distribute these average forces circumferentially across the cylinder based on the actuator element resolution instead, distribution coefficients for each actuator point m are computed for both the normal $d_N(\phi)$ and tangential $d_T(\phi)$ forces, using the previously computed azimuthal blade forces for each element in the spanwise direction, as shown in Eqs. (8) and (9):

$$d_N(\phi) = \frac{\sum_{i=i_{start}}^{i_{end}} F_{N,i} \Delta\phi R}{n \bar{F}_N} \quad (8)$$

$$d_T(\phi) = \frac{\sum_{i=i_{start}}^{i_{end}} F_{T,i} \Delta\phi R}{n \bar{F}_T} \quad (9)$$

where i_{start} and i_{end} correspond to the discrete bounds of $\phi - \frac{1}{2}\Delta\phi$ and $\phi + \frac{1}{2}\Delta\phi$, respectively, ϕ is the angle representing the azimuthal position of the actuator element centre. Hence, the computation of the body forces to be projected into the CFD domain becomes possible as in Eqs (10) and (11):

$$f_N(\phi) = \bar{F}_N d_N(\phi) \quad (10)$$

$$f_T(\phi) = \bar{F}_T d_T(\phi) \quad (11)$$

After computing the steady forces at each m_i element, these forces are projected into the steady CFD domain using a regularization kernel. A two-dimensional Gaussian smearing function is applied in tangential and normal directions, denoted using the subscript T and N , respectively, as in Eq. (12):

$$\eta(r_T, r_N) = \frac{1}{\beta_T \sqrt{\pi}} \exp \left[- \left(\frac{r_T}{\beta_T} \right)^2 \right] \frac{1}{\beta_N \sqrt{\pi}} \exp \left[- \left(\frac{r_N}{\beta_N} \right)^2 \right] \quad (12)$$

where r_T and r_N represent the distances between a given cell and the centre of the actuator element m_i , and β_T and β_N are width parameters that control kernel distribution. Fig. 3-b shows how these forces are projected, with β_T related to the hydrofoil thickness t and β_N related to an arc length proportional to $\Delta\varphi R$.

In the proposed AC model, certain sub-models and ad-hoc corrections, similar to those implemented in the ALM code referenced in (Melani et al., 2021c; Mohamed et al., 2024b), were employed to account for the second-order effects commonly observed in Darrieus turbines. To the authors' knowledge, the latter corrections are applied to the AC for the first time. These include:

- **Flow curvature effect:** several studies (Bianchini et al., 2015; Rainbird et al., 2015) show that the cycloidal motion of a Darrieus turbine blade deforms the surrounding streamlines, increasing their curvature relative to the freestream flow. This causes the blade to behave like a cambered hydrofoil, oriented outward from the turbine axis. To account for these effects, the polar data used in the present AC model was derived for a virtually cambered hydrofoil, as outlined in (Rainbird et al., 2015).
- **Dynamic stall and added mass effect:** The Berg model (Massé, 1981) is employed to account for dynamic stall effects, following the implementation in (Melani et al., 2021a). This model introduces a delay in the sampled angle of attack, which varies with the flow reduced frequency, capturing dynamic effects in the solution. Additionally, added mass terms are incorporated into the normal $C_{N,AM}$ and tangential $C_{T,AM}$ forces to account for the added mass effect, as in Eqs. (13) and (14):

$$C_{N,AM} = -\pi \cdot c \frac{dV_N}{dt} \frac{1}{8 V_{rel}^2} \quad (13)$$

$$C_{T,AM} = \pi \cdot c \frac{d\alpha}{dt} V_N \frac{1}{8 V_{rel}^2} \quad (14)$$

- **Tip loss correction:** The Dağ-Sørensen (DS) correction (Sørensen et al., 2014) is used to account for tip effects, as successfully implemented in the ALM code by the authors in (Melani et al., 2022; Mohamed et al., 2024b). The implementation of this correction in the ALM relies on the assumption of equivalence between an isotropic Gaussian smearing function and the Lamb-Oseen vortex, with the kernel width of the former being equivalent to the viscous radius of the latter. In the present AC model, on the other hand, the sampled spatial variation in the local velocity is treated as temporal variation, with a viscous radius equivalent to 0.25 chord, similar to the kernel width used in the ALM implementation. However, it is important to note that this is an approximation, given the different vortex systems created by both AC and ALM. Moreover, this formulation may introduce uncertainties when applied to blades with curved ends.

In summary, the algorithm of the present AC-VOF model operates as follows: at each time step, the flow field is sampled around the cylinder representing the turbine. The sampled local kinematic data is combined with the sub-models and ad-hoc corrections. This information is used to determine the equivalent lift and drag coefficient from tabulated polars and compute the corresponding instantaneous forces. The model then

calculates the average forces and projects them into the computational domain before advancing to the next time step, after which the algorithm repeats. The main concept is to integrate the steady AC model within the U-RANS solver. This approach enables the computation of mean turbine performance within a transient framework, allowing the use of large time steps.

As illustrated in Fig. 4, the AC algorithm shares a similar workflow with the ALM code recently presented by (Mohamed et al., 2024b). However, it differs in how flow kinematics are sampled, and turbine forces are projected, while the intermediate processes remain identical. A key distinction between the present model and the earlier ALM model lies in the time-stepping approach. In the ALM model, time-stepping is dictated by the ALM itself, where a small time-step is required. In contrast, the AC model is steady, and time-stepping is only required by the explicit solver of the VOF method. This enables the use of much larger time steps compared to the ALM, where the time step corresponds to an angular step of $\Delta\theta = 1^\circ$. For the AC model, several time steps, ranging from $\Delta\theta = 10^\circ$ to $\Delta\theta = 360^\circ$, were evaluated. The results of the AC model remained consistent across all cases; however, $\Delta\theta = 180^\circ$ was selected because stepping through a full cycle sometimes introduced numerical instability and convergence issues in the VOF solution.

Furthermore, a convergence analysis concluded that the model requires simulation times of approximately 50 revolutions to achieve a steady periodic condition.

3.2. CFD settings

In a hypothetical sloped channel with a constant, uniform cross-section, steady equilibrium between gravity and wall friction maintains a consistent water depth h_∞ and flow velocity V_∞ along the channel. Introducing HKTs disrupts this balance, causing a rise in the upstream water level and slowing the approach flow in front of the turbines. Downstream, the water level gradually returns to h_∞ . This is also known as "depth recuperation". The upstream water rise, measured as ΔH , reflects the difference with respect to the original water level.

In the present CFD model, the VOF method (Hirt and Nichols, 1981) was applied, which relies on the principle that multiple fluids cannot interpenetrate. It assigns a volume fraction to each fluid within each computational cell as a distinct variable. The volume fraction equation is solved for the secondary phase, while the volume fraction of the primary phase is calculated so that the total sum of volume fractions equals one. This method captures free-surface variations with the assumption of a horizontal, slope-free channel bed, previously validated in (Gauvin-Tremblay and Dumas, 2020; Mohamed et al., 2024b), using a constant surface tension of 0.072 N/m for the water surface. Under this setup, the water level can vary based on the flow field, with a fixed level at the outlet and flexible fluctuations at the inlet. Although this approach substantially reduces computational demand by simulating a shorter channel section, it limits the ability to calculate the distance required to reach depth recuperation. However, the upstream water rise ΔH can still be estimated, allowing the estimation of the power coefficient of the HKT array independently of the Froude number, which is influenced by the channel flow confinement.

Fig. 5 presents the computational domain and boundary conditions in the CFD model. The boundary conditions were set as follows: pressure inlet at the beginning of the channel, pressure outlets at the end of the channel and the top boundary, and no-slip for the channel walls. An initial water level of 6.52 m was set, gradually increasing during the solution to reach ΔH .

Regarding the simulation setup, it was chosen to be similar to the one used in the ALM-VOF model in (Mohamed et al., 2024b) to ensure an unbiased comparison. A pressure-based solver was used, employing the coupled algorithm, with a second-order upwind scheme for the spatial discretization of all transport equations and a second-order implicit scheme for the time discretization. Moreover, the $k-\omega$ SST model (Menter, 1993) was selected as the turbulence model due to its accuracy

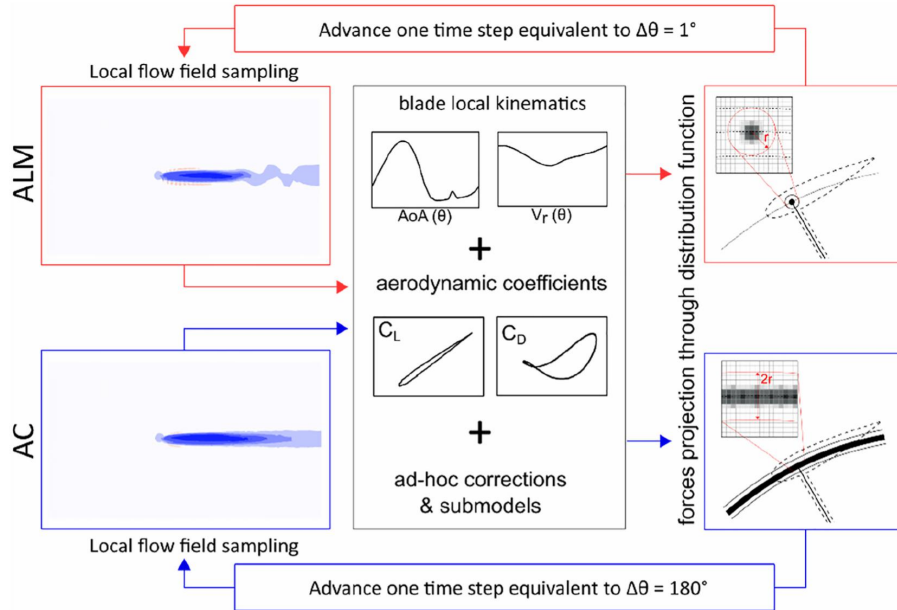


Fig. 4. Comparison between the workflow of the AC and that of the ALM.

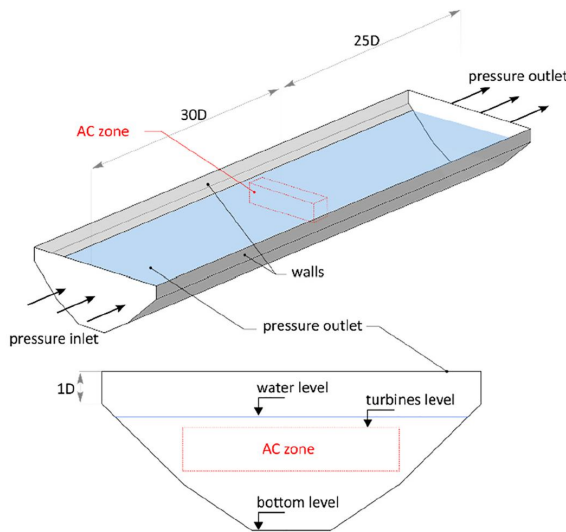


Fig. 5. CFD domain and boundary conditions.

and stability in unsteady flow conditions (Balduzzi et al., 2016). Finally, the residuals for all variables were set to 10^{-5} .

Both the domain size and structure were chosen based on a dedicated sensitivity analysis performed in (Mohamed et al., 2024b). The domain comprises an AC region and a flow field region, which extends 30D upstream and 25D downstream, respectively, where D represents the turbine diameter. Different from open-field simulations, accurate inflow estimation near the turbines in this model requires a sufficient distance for the upstream flow to fully develop.

4. Actuator cylinder tuning

This section presents the key findings from the sensitivity analysis of the proposed AC model. It begins by a mesh independency test to size the mesh in the AC region (section 4.1), then it highlights the importance of kinematic data sampling, specifically by comparing different measurement locations for the angle of attack and absolute inflow (Section 4.2). The analysis then addresses the influence of the regularization kernel size, which acts as a smoothing function for turbine forces (Section 4.3). Finally, it assesses the sensitivity of the model to variations in the number of actuator points m_i (Section 4.4).

4.1. Mesh independency test

As previously described, the computational domain is divided into the flow field and AC zones. The mesh comprises a combination of structured (AC region) and unstructured (flow field) grids. The mesh resolution in the flow field was already verified through a dedicated sensitivity analysis by (Mohamed et al., 2024b), ensuring sufficient refinement in the upstream and downstream regions to capture all the relevant flow scales of interest. However, a separate mesh independence test was conducted for the AC region.

In all meshes, the blade-wise distance along each rotor AC was discretized into 30 elements, with a biasing region near the rotor tip. Additionally, 15 inflation layers were applied near the channel wall to maintain a wall y^+ below 1, ensuring compliance with the requirements of the adopted turbulence model. The upstream and downstream flow field regions were divided using 60 and 50 divisions, respectively. Various mesh sizes were tested using different element sizes Δh , scaled with respect to the kernel width in the tangential and normal directions (β_T and β_N , respectively). Grid convergence was measured in terms of coefficient of determination R^2 as in Eq (15):

$$R^2 = 1 - \frac{\sum_{\theta=0^\circ}^{360^\circ} [C_T(\theta) - C_{T,finest}(\theta)]^2}{\sum_{\theta=0^\circ}^{360^\circ} [C_T(\theta) - C_{T,average}]^2} \quad (15)$$

where $C_T(\theta)$ is the instantaneous torque coefficient of the evaluated

mesh, $C_{T,finest}(\theta)$ corresponds to M03 in this case, and $C_{T,average}$ is the cyclic average torque coefficient.

Table 3 presents details of the three tested meshes in terms of element size in the AC region, along with their corresponding R^2 values, demonstrating that M02 achieves numerical independence with $R^2 > 99.99\%$. Fig. 6 reports the azimuthal variation of the torque coefficient for the three meshes, revealing that discrepancies in the coarse mesh primarily occur in the second quarter, with minor differences observed in the second half of the turbine cycle. The final mesh has a skewness of approximately 0.6 and an aspect ratio of about 30. Fig. 7 provides a visual representation of M02, detailing the mesh distribution in the AC and flow field zones.

4.2. Kinematic data sampling

For a proper modelling of Darrieus turbine performance, accurately sampling the local blade angle of attack α and chord-based Reynolds number Re_c is essential. Moreover, the local kinematic variations along the azimuthal path of a Darrieus turbines can be assumed equivalent to those experienced with time-stepping moving blades. In a steady AC approach, the flow field is sampled along the turbine diameter, representing flow changes relative to the azimuthal position. Unlike the ALM, which samples the velocity near the actuator line, the AC method samples the flow field along the cylinder, overlapping with the computational cells that project the forces. This requires careful attention to minimize noise.

A steady two-dimensional case was created for a stand-alone rotor to test the sensitivity of the sampled angle of attack α to the position at which it is sampled. Different sampling radii were tested as presented in Fig. 8, compared to the ALM transient solution. As it can be seen, the sampled angle of attack is influenced by the bound vorticities from the projected forces, with this influence increasing as the sampling radius approaches the turbine radius. The angle of attack results, sampled at the turbine radius, closely match the ALM solution. Moreover, the number of sampling points n_i is maintained as 360 to mimic the ALM sampling resolution with angular step size $\Delta\theta$ of 1° .

4.3. Regularization kernel

As noted earlier, the velocity is sampled from computational cells that intersect with those used to project the turbine forces. This means the accuracy of the AC model is sensitive to the size of the smearing function used for force projection.

Various kernel widths were evaluated at different tip-speed ratios (TSRs). Since the present model was developed on an already existing mesh, it was necessary to identify the minimum kernel size to avoid numerical instabilities or affect the quality of the sampled kinematic data. Fig. 9-a illustrates how smaller kernel sizes can lead to high levels of noise in the sampled angle of attack data using different sizes of $\beta_t/\Delta\phi$ with a constant $\beta_r/t = 1$ at $TSR = 2.5$. To assess the impact of different kernel width combinations in both the tangential and radial directions, the sum of squared errors (SSE) was computed by comparing the torque coefficient results from the AC model with those from the ALM model. The equation for SSE is as follows:

$$SSE = \sum_{\theta=0}^{360} (C_T(\theta)_{AC} - C_T(\theta)_{ALM})^2 \quad (16)$$

Table 3

Details of the evaluated meshes and their corresponding coefficient of determination (R^2).

Mesh	$\Delta h/\beta_T$	$\Delta h/\beta_N$	#elements	R^2
M01	1	1.4	5,317,444	99.94%
M02	0.5	0.7	6,625,040	99.99%
M03	0.25	0.35	8,285,079	–

Fig. 9-b shows the SSE contour for several AC model cases with varying kernel widths in both the tangential and radial directions. Results indicate that narrower kernel widths produce higher error values, primarily due to discontinuities in the angle of attack sampling across the turbine diameter when forces are projected over a confined area. Additionally, the operating TSR influences these errors, with lower TSRs resulting in higher SSE values when smaller kernel sizes are used. Based on these results, kernel widths $\beta_t/\Delta\phi R$ and β_r/t were chosen to be 1.

4.4. Number of actuator points m_i

Optimizing the resolution of actuator points m_i along the AC is essential, as it affects both the quality of the results and the computational cost. Reducing the number of actuator points decreases computational time as it minimises the loop size for each turbine, but it may impact the performance estimation as well as the accuracy of the estimated wake details. This part of the sensitivity analysis was conducted using three-dimensional simulations at various TSRs, evaluating different numbers of m_i points: 18, 36, 72, 90, and 180. The goal was to determine a threshold number of actuator points beyond which further increases do not significantly alter the AC solution.

To reach a solution independent from the number of m_i points used to construct the AC, the coefficient of determination R^2 is computed for each case using the estimated torque coefficient C_T , as shown in Eq (15). Fig. 10 shows the R^2 results for each case with different m_i values at TSRs of 2.26, 2.75, and 3.26. As can be seen, the lower the TSR, the less the solution depends on the number of m_i points. However, regardless of the operating TSR, an independent solution can be reached at $m_i = 90$, where for all cases R^2 is less than 99.99%.

5. Results and discussion

This section analyses and discusses the performance of the AC model under various operating conditions in comparison with the ALM and experimental measurements (Mohamed et al., 2024b). First, mean performance is evaluated in terms of average power coefficients. Next, the instantaneous performance of the models is compared, followed by an analysis of their respective impacts on channel hydraulics. Then, the resulting flow fields from both models are analysed. Finally, the computational time required for the AC model is discussed in relation to the ALM model.

5.1. Mean performance

The average power coefficient C_p of the stand-alone turbine and that of the second turbine in the array (T2) are presented across various TSRs in Fig. 11, as estimated by the AC and ALM models. The results for T2 from both models are compared with experimental measurements (Mohamed et al., 2024b). All results have been normalised by the maximum experimental power coefficient $C_{p,EXP MAX}$ due to industrial confidentiality. This normalisation, however, does not impact the comparative analysis.

The performance estimations for the stand-alone turbine are consistent between the AC and ALM models. Both showing models show a peak C_p at a TSR of 2.5 and a maximum percentage relative difference of 3.4% (AC relative to ALM). However, differences arise in the predictions of array performance. At low TSRs, the percentage difference between the models is relatively small (e.g., 6.8% at $TSR = 2.23$). As TSR increases, the AC model increasingly underestimates C_p compared to experimental measurements, with the relative difference between the two models reaching a maximum of 20.6% at $TSR = 3.14$. Notably, the ALM model provides predictions that align much more closely with experimental data overall. Conversely, while the AC model is closer to the experimental results at low TSRs, it fails to accurately capture the trend of C_p variation. At $TSR > 2.6$, where the array power augmentation effect becomes more prominent, the ALM model is much closer to the

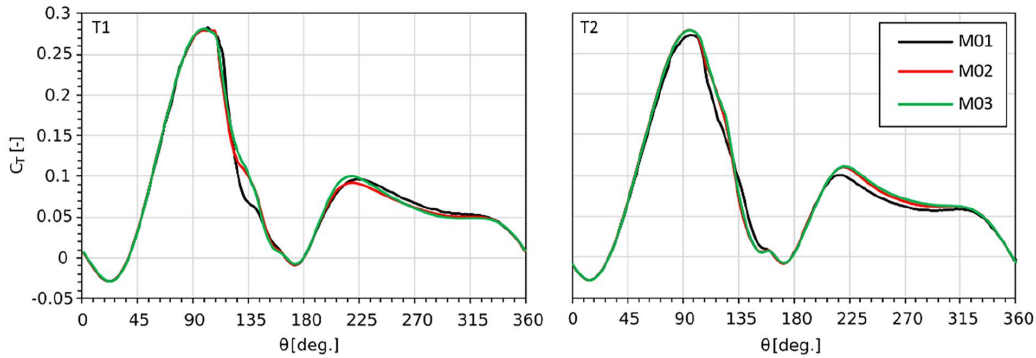


Fig. 6. Results of different meshes in terms of azimuthal torque coefficient.

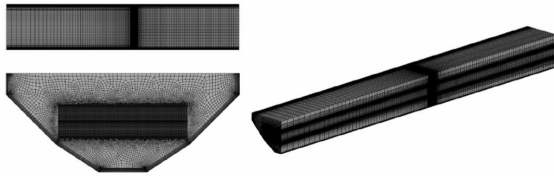


Fig. 7. Computational mesh.

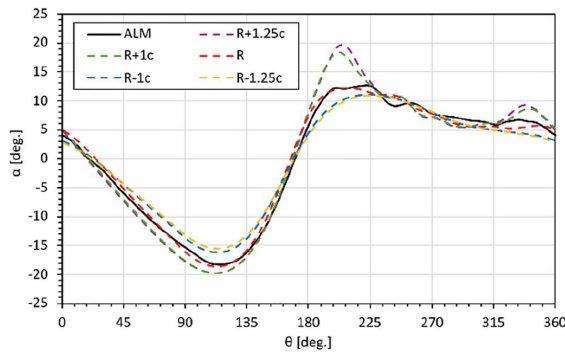


Fig. 8. Angle of attack α results of the AC compared to the ALM for different sampling radii.

experimental results compared to the AC model, where the maximum prediction error between the ALM and experimental measurements is 6%, compared to 19% for the AC model. Overall, the findings confirm that the ALM model is more effective at capturing the turbine-to-turbine interactions present in arrays. While this was to be expected due to the stronger simplifications introduced with the AC modelling, the accuracy of the AC is remarkable. Additionally, the accuracy degradation of both models at lower TSR is mainly due to the unsteady effects which are not fully captured using the Berg dynamic stall model. This has been investigated in more details in (Mohamed et al., 2022).

5.2. Instantaneous performance

To gain a better insight on the capabilities of the AC and ALM models in reproducing the blockage effects caused by closely deployed turbines, a comparative analysis was conducted. This analysis examined the azimuthal variation of the blade angle of attack α and the torque

coefficient C_T for both models, focusing on the stand-alone turbine in comparison with T1 and T2. Results are presented only for T1 and T2, as the problem under consideration is symmetric with respect to the centre of the channel.

Fig. 12 illustrates the variation in the angle of attack α at TSRs of 2.75 and 3.14. The increase in α in T1 and T2 due to turbine-to-turbine interactions is highlighted at the top of each graph to emphasise the differences between the two models in capturing this effect. For the stand-alone turbine, the α variations predicted by both models are close, although with some offsets, particularly during the first azimuthal quarter. In contrast, significant differences are observed in how the turbines in the array are influenced by one another in the two models. The AC model tends to predict α variations for turbines in the array that are closer to those of the stand-alone turbine, especially during the upwind half of the rotation. The ALM model, on the other hand, clearly shows an increase in α in regions of mutual interaction between adjacent turbines at both TSRs.

The latter observation highlights the basic reason for the underestimation of array performance in the AC model. Given that the primary mechanism driving power augmentation in Darrieus turbine arrays is the alteration of local flow kinematics, which leads to an increase in α in the areas of mutual flow interactions between adjacent turbines (Mohamed et al., 2023a). This observation suggests that the AC model, due to its inherent limitation in capturing unsteadiness, struggles to accurately represent this crucial effect. While the assumption of temporal variation in local kinematics in the ALM model can be considered analogous to the azimuthal variation in the AC model, the increase in α observed in arrays compared to the standalone turbine is not adequately captured by the AC model.

To analyse the influence of the latter differences between the ALM and AC models on turbine performance, the cyclic variation of the torque coefficient C_T is presented in Fig. 13 for T1 and T2, compared to that of the single rotor. The additional torque gained due to turbine-to-turbine interactions is highlighted for both turbines, along with the percentage increase in the power coefficient estimated by each model.

For the stand-alone turbine, a slight offset can be observed between ALM and AC estimations; however, the trend is nearly identical, and the average values closely match (see Fig. 11). Regarding the array performance, the two models produce significantly different results. The ALM clearly captures turbine-to-turbine interactions, showing a significant increase in the torque coefficient in regions of mutual interaction between adjacent turbines. This effect is more pronounced at a TSR of 3.14. Conversely, the AC model shows only slight increases in C_T during the upwind half of the rotation. During the downwind half, a significant increase in C_T is observed; while more apparent at a TSR of 3.14, it is still underestimated compared to the ALM.

At TSR = 2.75, the ALM estimates an increase in the overall array C_P due to turbine-to-turbine interactions of approximately 30%, while the

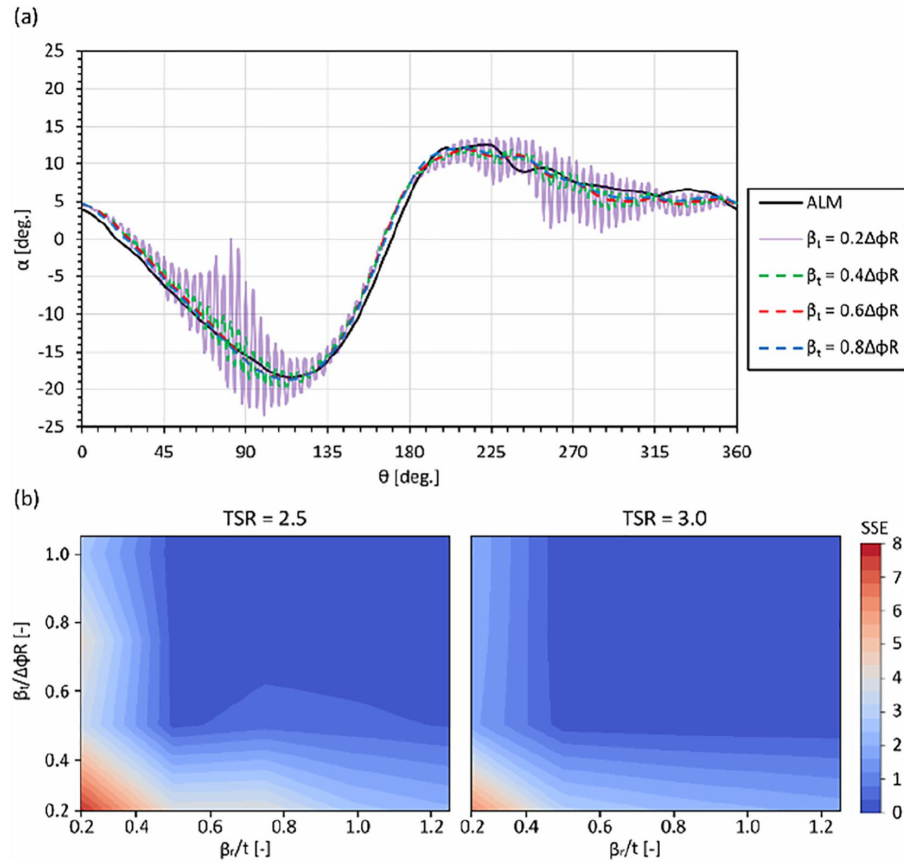


Fig. 9. (a) angle of attack results for different kernel width in tangential direction β_t , and (b) SSE results for different combinations of β_t and β_r .

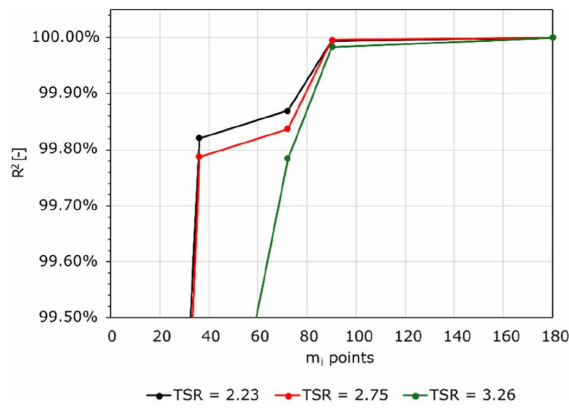


Fig. 10. Results of the R^2 for different number of actuator points at TSR 2.27, 2.75, and 3.26.

AC model estimates only about 11%. At TSR = 3.14, the ALM predicts a $\approx 49\%$ increase in the array C_p compared to the stand-alone turbine, whereas the AC model shows a $\approx 24\%$ increase. In general, the ALM provides predictions that align more closely with experimental measurements across both evaluated TSRs, capturing the power increase

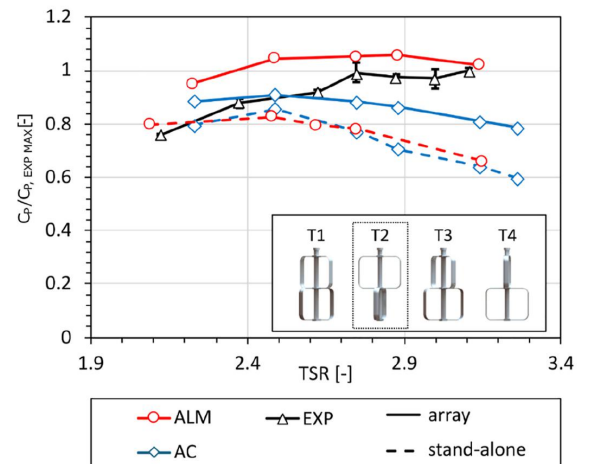


Fig. 11. comparison between experimental measurements, ALM, and AC in terms of average power coefficient normalised over the maximum experimental C_p for T2.

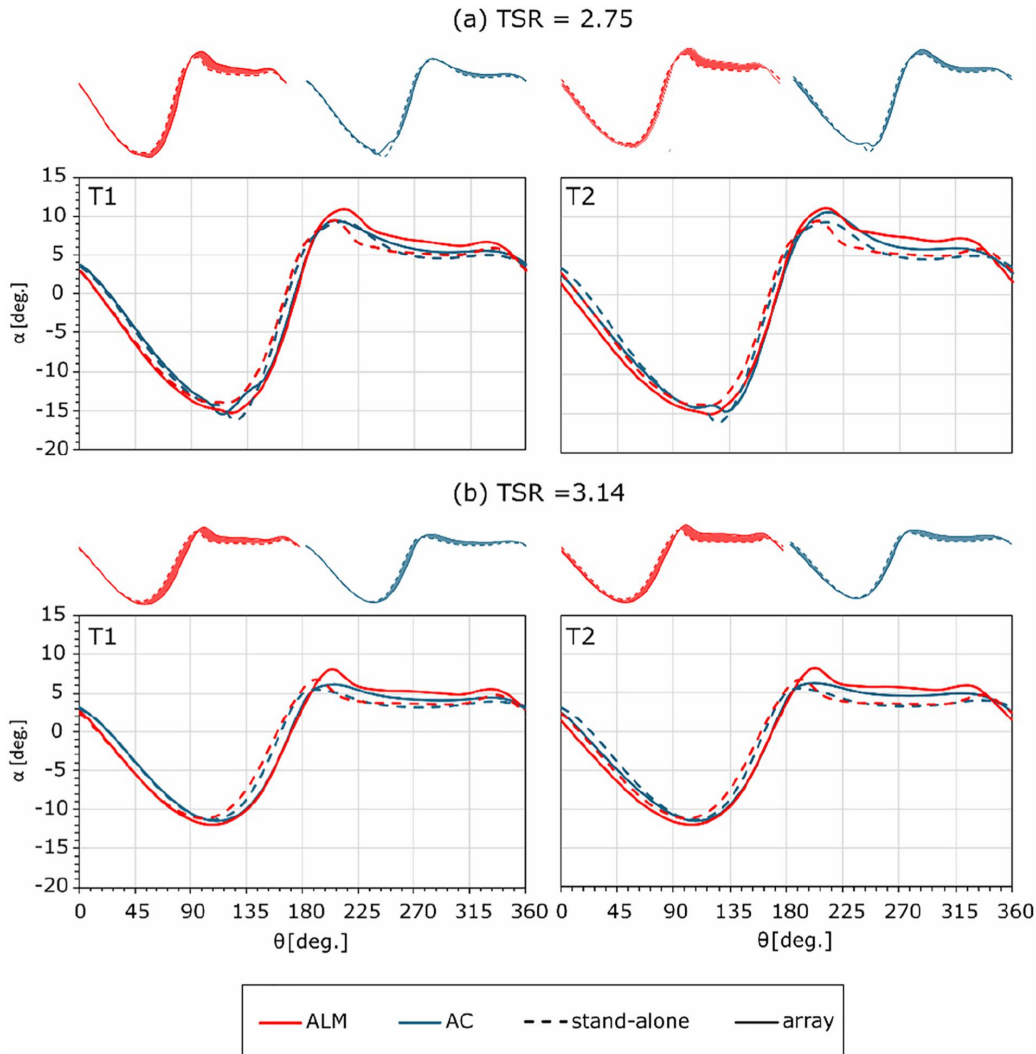


Fig. 12. Comparison between the angle of attack variation $\alpha(\theta)$ for stand-alone turbine compared to that of T1 and T2 of the array estimated by AC and ALM.

effect due to turbine-to-turbine interactions more accurately (see Fig. 11).

To evaluate the differences between the two models in estimating forces, Fig. 14 presents the average thrust coefficient results for the stand-alone turbine and the turbine array, as calculated by the AC and the ALM. The AC consistently underestimates the thrust compared to the ALM. However, despite this, the results for the stand-alone turbine appear to match in terms of the thrust variation trend across TSRs. In contrast, for the turbine array, the ALM predicts a steeper increase in thrust as the TSR increases.

Fig. 15 presents the normalised normal (left axis) and tangential (right axis) forces for turbines T1 and T2 at TSRs of 2.75 and 3.14, integrated over the entire blade span. The forces are normalised by $0.5\rho cV_\infty$, where V_∞ is the mainstream velocity. Since the stand-alone turbine predictions are nearly identical for the ALM and AC models, only the array results are presented to emphasise the differences between the models.

In general, the results show a close match between the two models

within the range $100^\circ < \theta < 200^\circ$. The peak tangential force, both upwind and downwind, decreases as the TSR increases because the normalisation is based on the mainstream velocity. At a TSR of 2.75, slight unsteadiness is observed during the second quarter, but these fluctuations progressively diminish with increasing TSR. By TSR 3.14, the tangential force no longer exhibits pronounced load fluctuations. The normal force, in contrast, increases in the upwind half as TSR rises, while remaining nearly constant across all TSRs in the downwind half. Accurate prediction of the normal force is particularly important, as it plays a significant role in shaping the wake flow downstream of the turbine array.

5.3. Channel hydraulics

In this section, the AC and ALM models are compared based on their ability to capture the two-way interaction between the HKT array and the channel, referred to as turbine-to-channel interaction to distinguish it from the turbine-to-turbine interaction observed within arrays. The

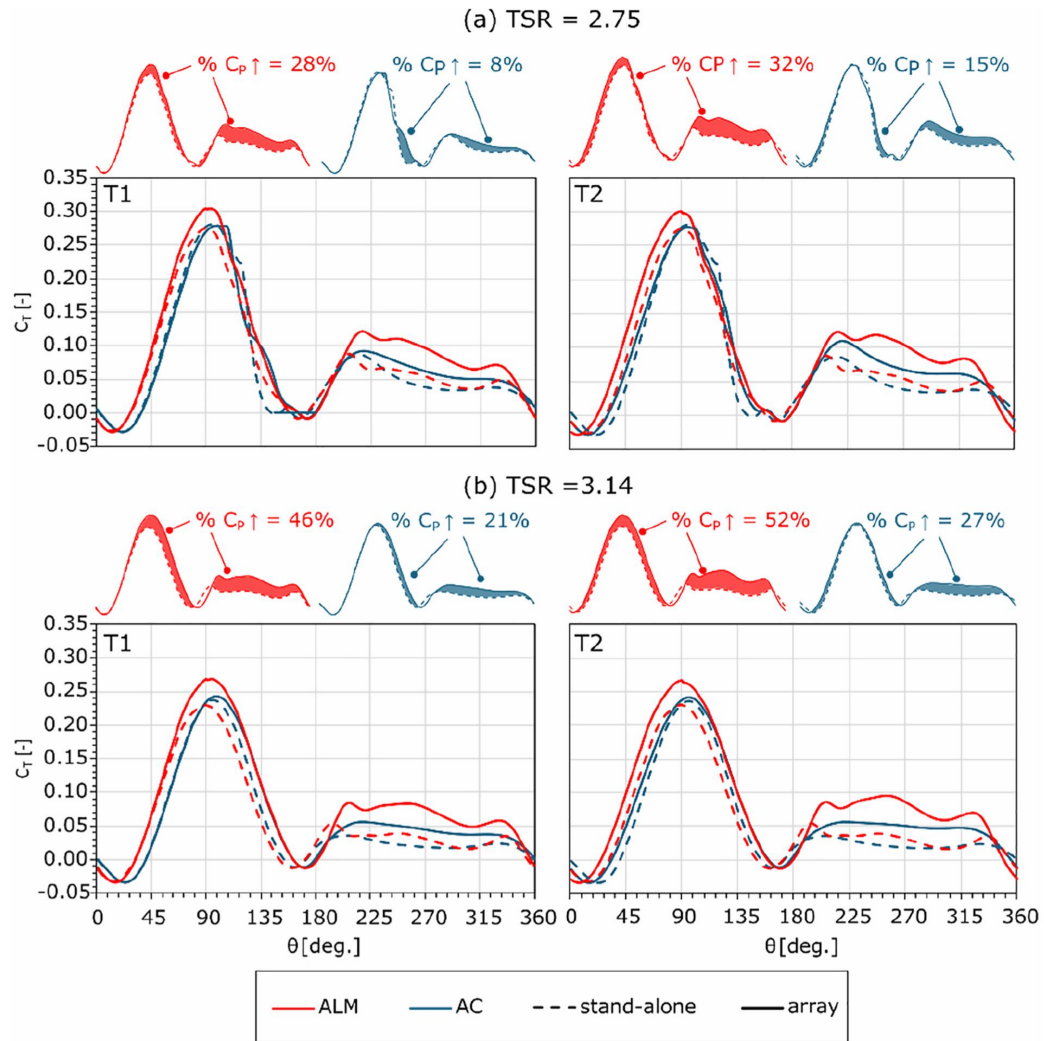


Fig. 13. Comparison between the torque coefficient $C_T(\theta)$ for stand-alone turbine compared to that of T1 and T2 of the array estimated by AC and ALM with percentage increase of power coefficient C_p for each turbine.

analysis begins by examining the variation in the average water level upstream of the array across different TSRs. Subsequently, the water level profiles along various sections of the channel are presented for both models to provide deeper insight into how they reproduce the hydraulic impact of the turbine array on the channel.

The upstream water rise, ΔH , is reported in Fig. 16, calculated based on the average channel cross-sectional area at $X/D = -1$ for both the AC and ALM solutions, and compared with experimental measurements (Mohamed et al., 2024b). Experimental data show that ΔH remains nearly constant with respect to turbine rotational speed up to $TSR \approx 2.5$. Beyond this point, ΔH increases sharply at $TSR = 2.75$, before declining and then rising again. The numerical models predict different trends: the AC solution shows an almost linear increase in ΔH with TSR, while the ALM solution peaks at $TSR = 2.75$ and then decreases. For the ALM solution, the decrease in ΔH corresponds to a decrease in the power coefficient. In contrast, the AC solution exhibits a continuous increase in ΔH , even as C_p steadily decreases (see Fig. 11).

Fig. 17 illustrates the variation in water level estimated by the AC

and ALM models for both stand-alone and array setups at $TSR = 2.75$. The results are presented across the streamwise distance $-20 < X/D < 20$ at different XZ planes and various longitudinal Y/D distances. Due to the geometric symmetry, only one half of the channel is shown.

As observed, both models show excellent agreement, particularly for the stand-alone turbine, demonstrating similar hydraulic behaviour attributable to array blockage. For the array setup, the impact of turbine-to-channel interactions is more pronounced due to the greater blockage effect imposed by the array compared to the stand-alone turbine. This includes a significant rise in water level upstream of the array, followed by a sharp drop around the turbines, and then a gradual return to a nearly constant value downstream.

The AC model slightly overestimates the water level across most of the channel compared to the ALM, with the discrepancy diminishing near the channel walls. For the array cases, the AC solution generally exaggerates turbine-to-channel interactions, leading to a more pronounced increase in water level compared to the ALM solution. Although the differences in water level are minor (see Fig. 16),

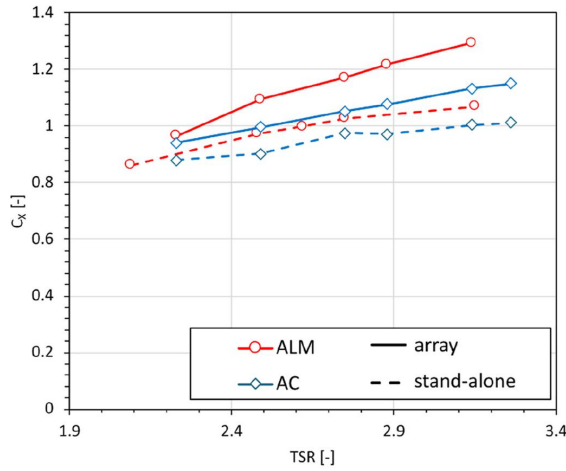


Fig. 14. Comparison between the average thrust coefficient for stand-alone turbine and array (averaged) estimated by the proposed AC model and the ALM.

accurately estimating this rise is crucial. It significantly influences the calculation of the corrected power coefficient, particularly when accounting for the blockage effect caused by the upstream water level increase. On the other hand, the agreement seen in Fig. 17 is thought to be more than acceptable in the case numerical simulations are used for estimating the impact of the turbines on the channel hydraulics far from the traverse (which is often the case in industrial projects).

5.4. Flow field

To assess the differences between the ALM and the AC in estimating

the flow field around the array, the flow upstream and downstream the turbines were sampled and compared to that of the ALM and experimental measurements in (Mohamed et al., 2024b).

Fig. 18-a illustrates the time-averaged vertical velocity profiles measured upstream of each turbine at $X/D = -2.96$, along with the corresponding results from the AC and ALM simulations. Both numerical methods predict an almost uniform velocity across the cross-sectional area, while experimental measurements show slight variations in velocity. Upstream, the velocity deficit is slightly greater in all AC cases compared to the ALM, attributed to the higher upstream rise predicted by the AC model. Nevertheless, the numerical and experimental profiles show good overall agreement, indicating comparable inflow conditions. Minor discrepancies near the walls are observed for both methods, likely due to the numerical boundary conditions and uncertainties in experimental measurements near the walls.

Fig. 18-b depicts the downstream velocity deficit for each turbine at

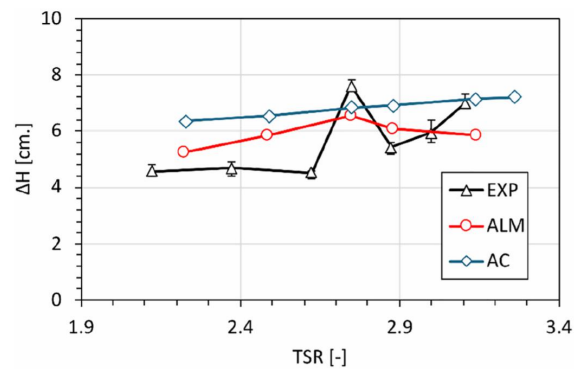


Fig. 16. comparison between experimental measurements, ALM, and AC in terms of upstream water rise ΔH .

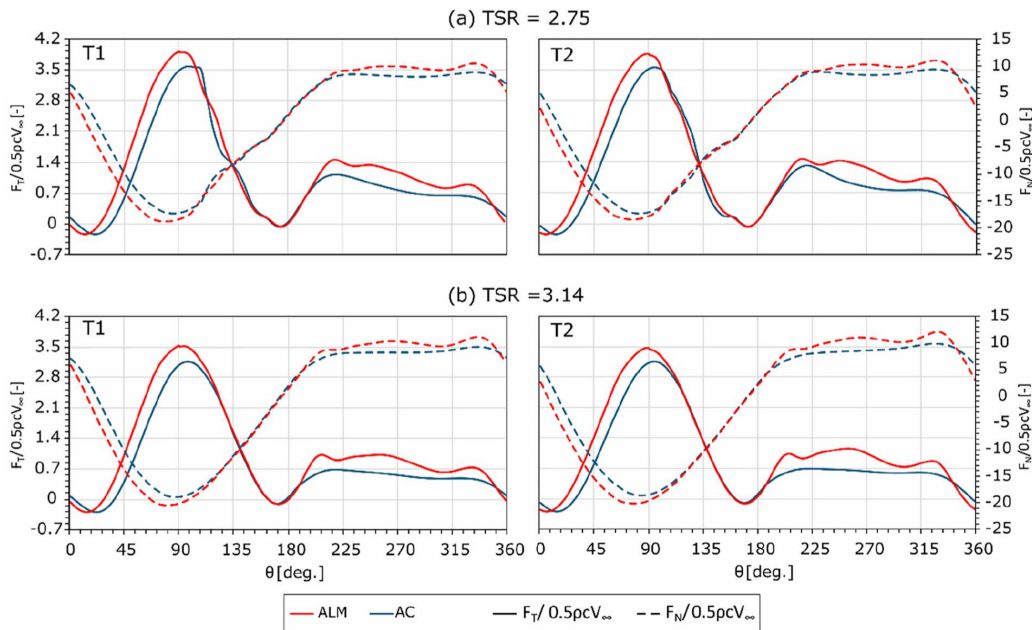


Fig. 15. Comparison between the variation of normalised tangential $F_T/0.5pcV_\infty$ and normal $F_N/0.5pcV_\infty$ forces estimated by the proposed AC model and the ALM.

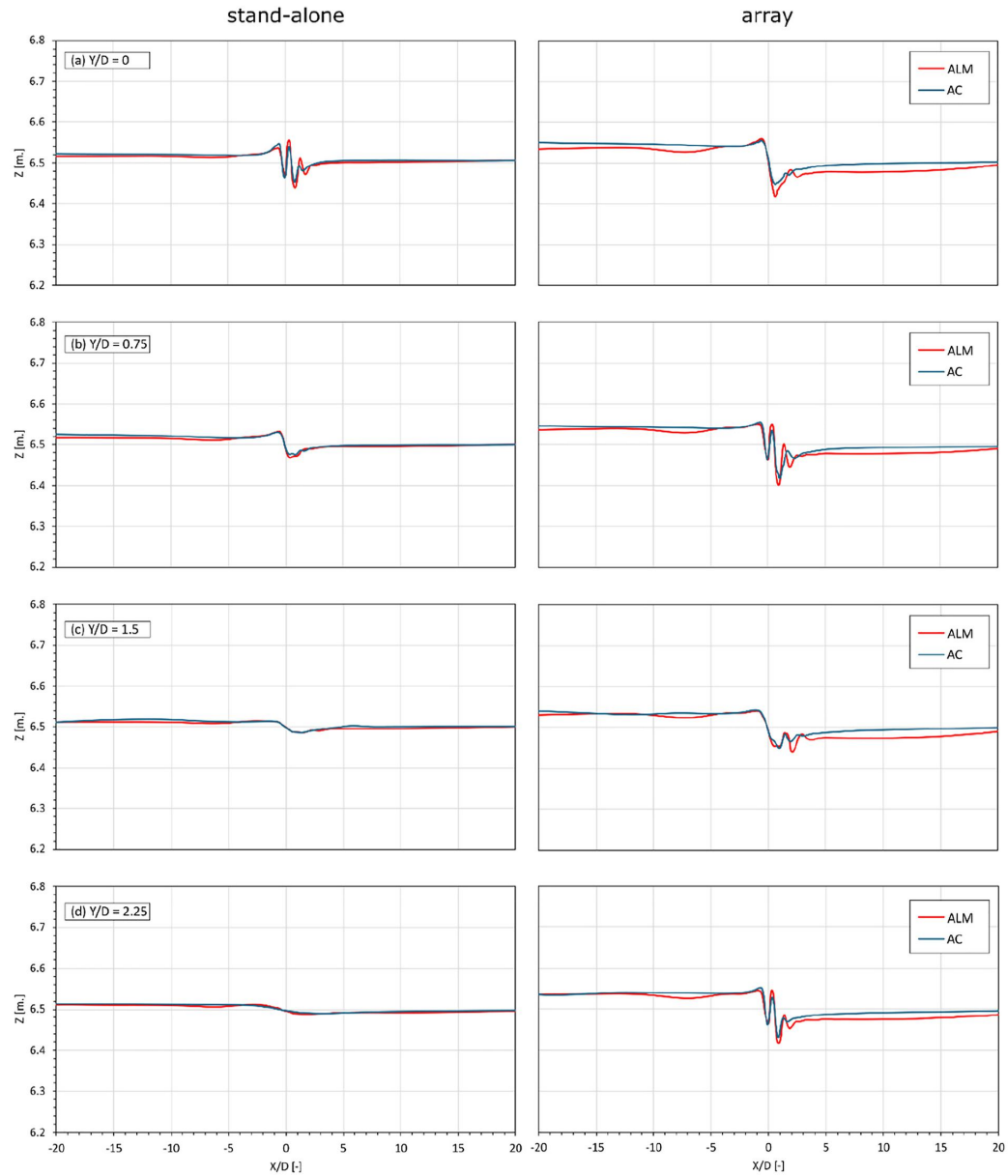


Fig. 17. variation in water level at different lateral distances, comparison between ALM and AC.

$X/D = 6$. A clear velocity deficit is observed in the wake of the turbines, along with regions of acceleration near the sides and the channel bottom, caused by flow interactions near the channel walls and the resulting blockage effect. The velocity deficit predicted by the AC model is notably higher than that of the ALM, particularly downstream of the lower rotor. The turbines occupy the Z -range from 3.2 m to 6.2 m. While the velocity profiles downstream of the upper rotor show good agreement between the AC, ALM, and experimental results, greater divergence is observed downstream of the lower rotor, with the ALM results aligning more closely with experimental measurements. Discrepancies near the walls are again evident, likely due to challenges in obtaining

accurate experimental measurements near the boundaries.

Fig. 19 reports the dimensionless velocity profile in the streamwise direction for the stand-alone turbine, as well as for T1 and T2 in the array. The velocity is sampled along a line passing through the turbine midspan, i.e., at $Z = 4.7$ m. For the stand-alone rotor, the difference between the upstream velocity estimated by the ALM and AC is minimal. However, in the array case, a significant difference is observed between the upstream velocities predicted by the ALM and AC. In general, for all cases, the upstream velocity is lower for the AC simulations, which corresponds to an overestimated blockage due to turbine-to-channel interactions, as noticed in section 5.3, causing a higher upstream

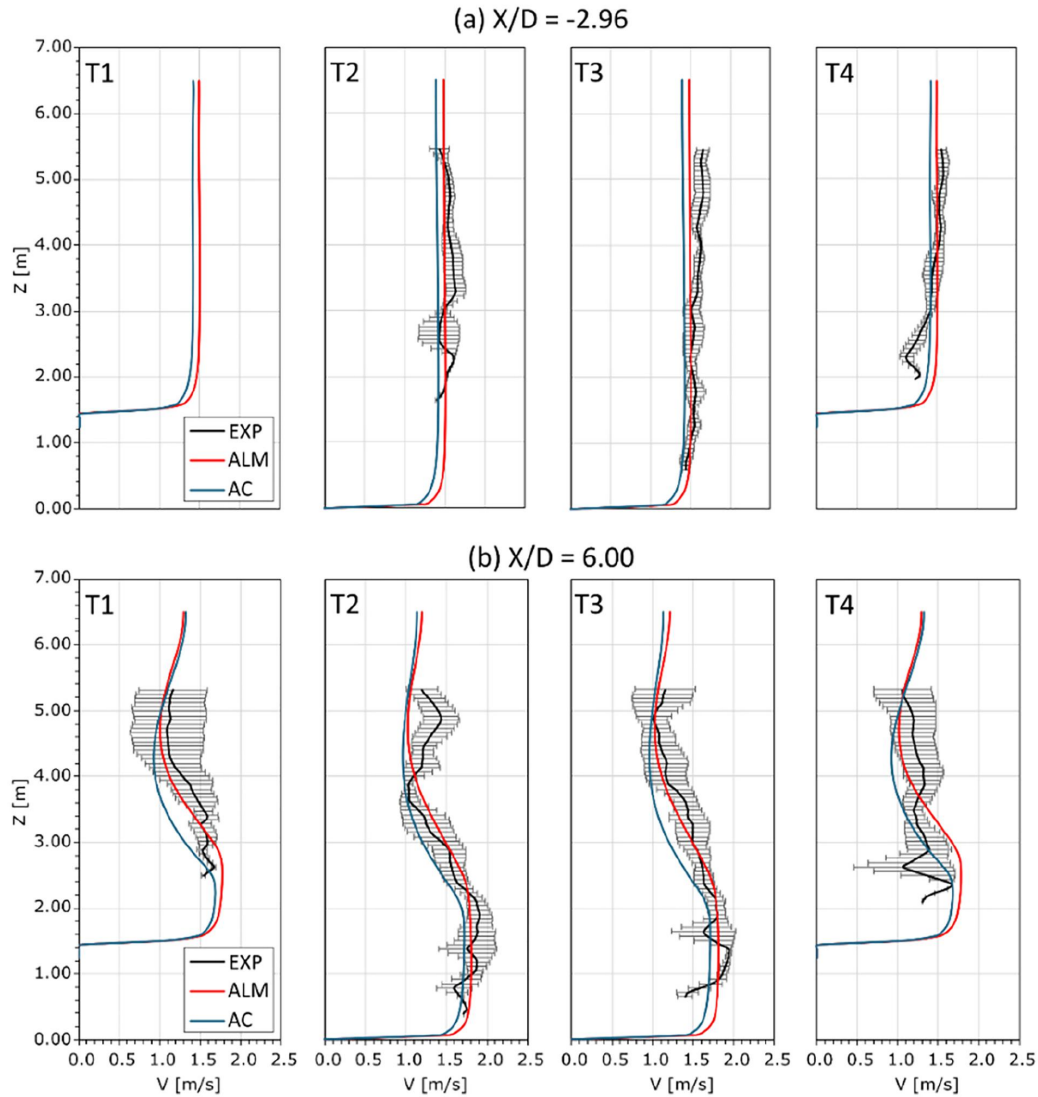


Fig. 18. Comparison between longitudinal velocity profiles from the AC and ALM models with experimental data upstream and downstream of the turbine array at $TSR = 2.75$.

water rise in the AC cases.

To assess the flow field results from both models, Fig. 20 presents velocity contours for the stand-alone turbine and the turbine array as predicted by the AC and ALM models, made non-dimensional dividing by the inlet velocity V_∞ . The results are shown from a top view of the turbine mid-plane.

For the stand-alone turbine, in the absence of turbine-to-turbine interactions and with minimal turbine-to-channel interaction, the flow fields produced by the AC and ALM models align closely, both displaying nearly identical velocity deficit profiles in the wake. In both models, the wake velocity recovers to approximately 80% of the free-stream velocity at $X/D \approx 9$.

Regarding the wake flow field around the array, the shape of the flow field offers valuable insights into how the AC and ALM models capture turbine-to-channel interactions. According to the ALM results, a

deceleration zone forms upstream of the turbines, resembling the behaviour observed in front of the stand-alone turbine. Additionally, a pronounced flow acceleration zone appears along the sides of the array and within the mutual interaction zones between adjacent turbines. In contrast, the AC model predicts a more pronounced blockage effect due to turbine-to-channel interactions, with the velocity deceleration zone extending further upstream of the rotor. However, the cross-stream acceleration zones caused by turbine-to-turbine interactions, which are well-represented in the ALM solution, are underestimated in the AC solution. In the AC model, only small regions of accelerated flow are observed near the sides of the array. As for the wake shape behind the array, both models exhibit good agreement, though the AC solution shows a more pronounced velocity deceleration further downstream at $X/D > 12.5$.

Fig. 21 compares the elevation view of the dimensionless velocity

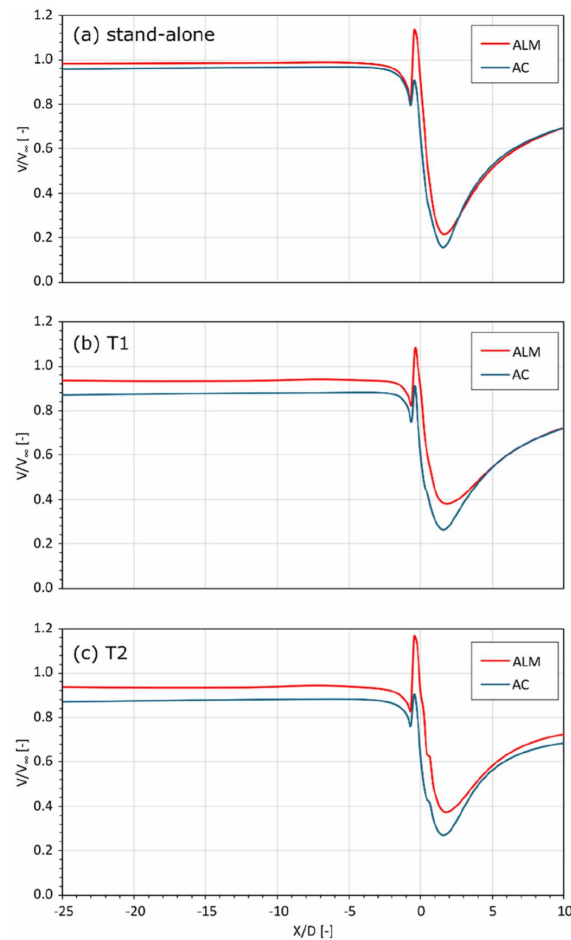


Fig. 19. Comparison between streamwise velocity profiles from the AC and ALM models at turbine midspan and $TSR = 2.75$.

contours for the stand-alone turbine as well as turbines T1 and T2 in the array, as predicted by the AC and ALM models. For the stand-alone turbine, the results show minimal variations between the AC and ALM models, consistent with earlier findings.

In the array results, however, the AC model predicts a more pronounced blockage effect due to turbine-to-channel interactions for T1 and T2 compared to the ALM. This is evident in the wider deceleration zone upstream of the array, which corresponds to the higher upstream water level highlighted in Section 5.3. Additionally, an acceleration zone forms beneath the turbines in the array, with velocities reaching approximately $1.2V_{\infty}$. The AC model underestimates this acceleration zone, showing a smaller acceleration region that is nearly identical for both T1 and T2. In contrast, the ALM model predicts a stronger and more distinct acceleration zone beneath T2.

5.5. Computational cost

Although the quasi-steady AC model does not produce results as realistic as the ALM, either in capturing the power augmentation due to turbine placement in an array or in representing the blockage effects caused by channel walls and the free water surface, it provides reasonable estimations of HKT performance and their impact on the channel,

largely within the uncertainty of typical industrial measurements. The primary advantage of the AC model lies in its lower computational cost compared to the ALM, making it a practical tool for preliminary design and optimisation, particularly in the industrial sector, where rapid analysis is crucial.

To assess the computational efficiency of the AC model, Table 4 presents the average computational time required for both AC and ALM simulations, performed using 208 processor cores. The simulations were executed on an AMD® EPYC® 2.65 GHz processor. While the AC model takes longer to achieve a converged solution, its total clock time is an order of magnitude lower than that of the ALM. It can be observed that the AC model scales differently compared to the ALM with respect to the number of turbines: the time required to model a 4-turbine array is nearly four times the time required for a stand-alone turbine. In contrast, the ALM does not exhibit this direct proportionality, as the time required for the array is approximately double that required for a stand-alone turbine.

Overall, comparing the AC to the ALM clearly demonstrates that the AC model requires an order of magnitude less computational time than the ALM in all cases, highlighting the advantages of the AC model in terms of computational efficiency.

6. Conclusions

A quasi-steady AC model was developed within the U-RANS framework of Ansys Fluent to simulate HKT arrays under realistic conditions, incorporating the VOF method to account for free water surface. Key model parameters, such as flow field sampling, kernel size, and resolution of actuator points, were tuned during the development process. A detailed code-to-code comparison was conducted between the AC and ALM models, focusing on assessing the capabilities of the proposed low-cost model in estimating channel blockage effects and power augmentation in arrays compared to a stand-alone turbine. Experimental data were included where available to serve as a reference for validation. The key findings of the study are summarised as follows:

- For stand-alone turbines, the AC and ALM models showed strong agreement, with minimal turbine-to-channel interactions observed in the single-turbine case due to low blockage.
- In array setups, the ALM demonstrated superior accuracy compared to the AC in estimating HKT performance, closely aligning with experimental measurements.
- The AC model tended to overestimate the channel blockage effect, predicting a higher upstream water level and lower upstream inflow velocity compared to the ALM.
- While the AC reproduced a slight power gain from closely deployed turbines, it showed only a small increase in efficiency compared to a stand-alone turbine with respect to the ALM, indicating underestimation of the turbine-to-turbine blockage, which is attributed to its unsteady nature.
- Despite discrepancies between the AC and ALM models, the AC model can still produce fairly good estimations of azimuthally dependent quantities, making it suitable for rapid estimation of instantaneous forces.
- The AC model demonstrated a remarkable reduction in computational time, requiring only about 7% of the time for stand-alone turbine simulations and 17% for array simulations compared to an equivalent ALM simulation. This significant efficiency makes the AC model an ideal choice for studies constrained by limited computational resources.

Overall, the AC model, despite its limitations in modelling turbine-to-channel and turbine-to-turbine interactions compared to the ALM, has proven to be a practical and computationally efficient tool, offering a reasonable trade-off between accuracy and computational demand. This makes it particularly suitable for analysing Darrieus turbine arrays in

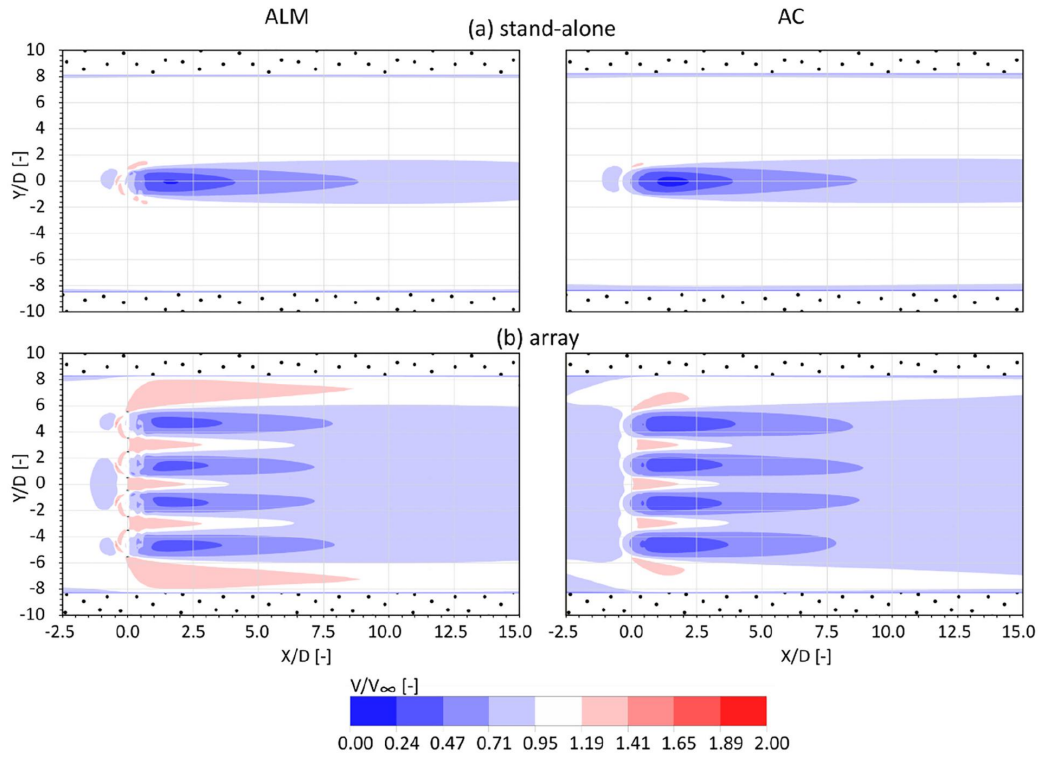


Fig. 20. Comparison between the ALM and the AC in terms of dimensionless velocity V/V_∞ contours of the array at $TSR = 2.75$ at turbine mid-plane.

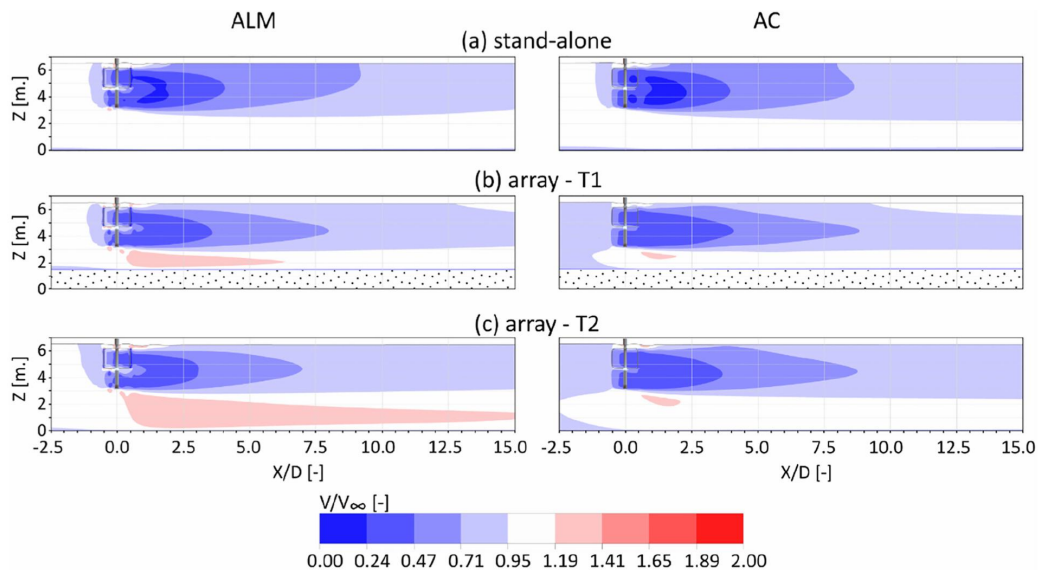


Fig. 21. Comparison between the ALM and the AC in terms of dimensionless velocity V/V_∞ contours of the array at $TSR = 2.75$ elevation view for stand-alone turbine, T1 and T2.

industrial applications where rapid, resource-efficient analysis is a priority. Future work should be undertaken to further compare the proposed model with other models, as well as to improve the capabilities of

the model through the integration of a more reliable dynamic stall model that better represents the influence of unsteadiness on aerodynamic loads.

Table 4
Comparison in terms of computational cost between AC and ALM.

Value	AC		ALM	
	Stand-alone	array	Stand-alone	array
Total simulation time [h/core]	423	1824	6132	10,578
Simulated flow time [s]	≈70		≈30	

CRedit authorship contribution statement

Omar S. Mohamed: Writing – original draft, Visualization, Validation, Software, Methodology, Formal analysis, Data curation, Conceptualization. **Pier Francesco Melani:** Writing – review & editing, Visualization, Supervision, Software, Methodology, Formal analysis, Data curation. **Giuseppe Soraperra:** Writing – review & editing, Supervision, Project administration. **Alessandra Brighenti:** Writing – review & editing, Supervision, Project administration. **Giovanni Ferrara:** Writing – review & editing, Supervision. **Vittorino Betti:** Writing – review & editing, Supervision, Funding acquisition. **Francesco Balduzzi:** Writing – review & editing, Supervision, Methodology. **Alessandro Bianchini:** Writing – review & editing, Supervision, Project administration, Methodology, Funding acquisition, Conceptualization.

Declaration of competing interest

The authors declare that they have no known competing financial interests or personal relationships that could have appeared to influence the work reported in this paper.

References

- Balduzzi, F., Bianchini, A., Maleci, R., Ferrara, G., Ferrari, L., 2016. Critical issues in the CFD simulation of Darrieus wind turbines. *Renew. Energy* 85, 419–435. <https://doi.org/10.1016/j.renene.2015.06.048>.
- Balduzzi, F., Melani, P.F., Soraperra, G., Brighenti, A., Battisti, L., Bianchini, A., 2021. Some design guidelines to adapt a Darrieus vertical axis turbine for use in hydrokinetic applications. *E3S Web Conf.* 312, 08017. <https://doi.org/10.1051/e3sconf/202131208017>.
- Bianchini, A., Balduzzi, F., Rainbird, J.M., Peiro, J., Graham, J.M.R., Ferrara, G., Ferrari, L., 2015. An experimental and numerical assessment of airfoil polars for use in Darrieus wind turbines: Part 1 — flow curvature effects, ASME turbo expo 2015. *Turbine Tech. Conf. Exposition*. <https://doi.org/10.1115/GT2015-42284>.
- Cacciali, L., Battisti, L., Dell'Anna, S., 2022. Free surface double actuator disc theory and double multiple Streamtube model for in-stream Darrieus hydrokinetic turbines. *Ocean Eng.* 260, 112017. <https://doi.org/10.1016/j.oceaneng.2022.112017>.
- Cacciali, L., Battisti, L., Dell'Anna, S., 2023a. Backwater assessment for the energy conversion through hydrokinetic turbines in subcritical prismatic canals. *Ocean Eng.* 267, 113246. <https://doi.org/10.1016/j.oceaneng.2022.113246>.
- Cacciali, L., Battisti, L., Dell'Anna, S., 2023b. Multi-array design for hydrokinetic turbines in hydropower canals. *Energies* 16, 2279. <https://doi.org/10.3390/en16052279>.
- Ferreira, C.S., Madsen, H.A., Barone, M., Roscher, B., Deglaire, P., Arduin, I., 2014. Comparison of aerodynamic models for vertical Axis wind turbines. *J. Phys. Conf. Ser.* 524, 012125. <https://doi.org/10.1088/1742-6596/524/1/012125>.
- Gauvin-Tremblay, O., Dumas, G., 2020. Two-way interaction between river and deployed cross-flow hydrokinetic turbines. *J. Renew. Sustain. Energy* 12, 034501. <https://doi.org/10.1063/5.0004492>.
- Guerrero, M., Conevski, S., Cavalieri, I., Schippa, L., Ruther, N., 2023. Influence of hydropower propellers on open-channel flow. *J. Hydraul. Eng.* 149, 04023047. <https://doi.org/10.1061/JHEND8.HYENG-13417>.
- Hirt, C.W., Nichols, B.D., 1981. Volume of fluid (VOF) method for the dynamics of free boundaries. *J. Comput. Phys.* 39, 201–225. [https://doi.org/10.1016/0021-9991\(81\)90145-5](https://doi.org/10.1016/0021-9991(81)90145-5).
- Holanda, P. da S., Blanco, C.J.C., Mesquita, A.L.A., Brasil Junior, A.C.P., de Figueiredo, N.M., Macêdo, E.N., Secretan, Y., 2017. Assessment of hydrokinetic energy resources downstream of hydropower plants. *Renew. Energy* 101, 1203–1214. <https://doi.org/10.1016/j.renene.2016.10.011>.
- Il Canale Biffis - Claudio Malini - Libro - Sometti - Acque e bonifiche | IBS, last access: 22 February 2024. <https://www.ibs.it/canale-biffis-libro-claudio-malini/e/9788874956760>.
- Kirke, B., 2019. Hydrokinetic and ultra-low head turbines in rivers: a reality check. *Energy Sustain. Dev.* 52, 1–10. <https://doi.org/10.1016/j.esd.2019.06.002>.
- Madsen, H., 1982. The actuator cylinder - a flow model for vertical Axis wind turbines. <https://doi.org/10.13140/RG.2.1.2512.3040>.
- Madsen, H.A., Paulsen, U.S., Vitae, L., 2014. Analysis of VAWT aerodynamics and design using the Actuator Cylinder flow model. *J. Phys. Conf. Ser.* 555, 012065. <https://doi.org/10.1088/1742-6596/555/1/012065>.
- Marten, D., Bianchini, A., Pechlivanoglou, G., Balduzzi, F., Nayeri, C.N., Ferrara, G., Paschereit, C.O., Ferrari, L., 2016. Effects of airfoil's polar data in the stall region on the estimation of Darrieus wind turbines performance. *J. Eng. Gas Turbines Power* 139. <https://doi.org/10.1115/1.4034326>, 022606–9.
- Massé, B., 1981. Description de deux programmes d'ordinateur pour le calcul des performances et des charges aérodynamiques pour les éoliennes à axe vertical.
- Melani, P.F., Balduzzi, F., Bianchini, A., 2021a. A robust procedure to implement dynamic stall models into actuator line methods for the simulation of vertical-Axis wind turbines. *J. Eng. Gas Turbines Power* 143. <https://doi.org/10.1115/1.4051909>.
- Melani, P.F., Balduzzi, F., Ferrara, G., Bianchini, A., 2021b. Development of a desmodromic variable pitch system for hydrokinetic turbines. *Energy Convers. Manag.* 250, 114890. <https://doi.org/10.1016/j.enconman.2021.114890>.
- Melani, P.F., Balduzzi, F., Ferrara, G., Bianchini, A., 2021c. Tailoring the actuator line theory to the simulation of Vertical-Axis Wind Turbines. *Energy Convers. Manag.* 243, 114422. <https://doi.org/10.1016/j.enconman.2021.114422>.
- Melani, P.F., Balduzzi, F., Bianchini, A., 2022. Simulating tip effects in vertical-axis wind turbines with the actuator line method. *J. Phys. Conf. Ser.* 2265, 032028. <https://doi.org/10.1088/1742-6596/2265/3/032028>.
- Menter, F.R., 1993. Zonal two equation kappa-omega turbulence models for aerodynamic flows. In: *AIAA Fluid Dynamics Conference*. NTRS Author Affiliations: Eloquent Corp.NTRS Report/Patent Number: AIAA Paper 93-2906NTRS Document ID: 19960044572NTRS Research Center: Ames Research Center (ARC), Orlando, FL.
- Menter, F.R., Smirnov, P.E., Liu, T., Avancha, R., 2015. A one-equation local correlation-based transition model. *Flow Turbul. Combust.* 95, 583–619. <https://doi.org/10.1007/s10494-015-9622-4>.
- Mohamed, O.S., Melani, P.F., Balduzzi, F., Ferrara, G., Bianchini, A., 2022. An insight on the key factors influencing the accuracy of the actuator line method for use in vertical-axis turbines: limitations and open challenges. *Energy Convers. Manag.* 270, 116249. <https://doi.org/10.1016/j.enconman.2022.116249>.
- Mohamed, O.S., Melani, P.F., Balduzzi, F., Ferrara, G., Bianchini, A., 2023a. An insight on the physical mechanisms responsible of power augmentation in a pair of counter-rotating Darrieus turbines. *Energy Convers. Manag.* 284, 116991. <https://doi.org/10.1016/j.enconman.2023.116991>.
- Mohamed, O.S., Melani, P.F., Papi, F., Balduzzi, F., Bianchini, A., 2023b. Is the Actuator line method able to reproduce the interaction between closely-spaced Darrieus rotors? a critical assessment on wind and hydrokinetic turbines. *Energy Convers. Manag.* 293. <https://doi.org/10.1016/j.enconman.2023.117473>.
- Mohamed, O.S., Melani, P.F., Soraperra, G., Brighenti, A., Balduzzi, F., Bianchini, A., 2024a. Selection criteria for rotor number and rotational direction in arrays of closely spaced Darrieus turbines. *J. Fluid Eng.* 1–32. <https://doi.org/10.1115/1.4067007>.
- Mohamed, O.S., Melani, P.F., Soraperra, G., Brighenti, A., Ferrara, G., Betti, V., Schippa, L., Guerrero, M., Balduzzi, F., Bianchini, A., 2024b. Three-dimensional CFD-ALM-VOF modeling of hydrokinetic turbines in realistic open-channel conditions. *Ocean Eng.* 313, 119411. <https://doi.org/10.1016/j.oceaneng.2024.119411>.
- Niebuhr, C.M., van Dijk, M., Neary, V.S., Bhagwan, J.N., 2019. A review of hydrokinetic turbines and enhancement techniques for canal installations: technology, applicability and potential. *Renew. Sustain. Energy Rev.* 113, 109240. <https://doi.org/10.1016/j.rser.2019.06.047>.
- Ning, A., 2016. Actuator cylinder theory for multiple vertical axis wind turbines. *Wind Energy Sci.* 1, 327–340. <https://doi.org/10.5194/wes-1-327-2016>.
- Paraschivoiu, I., 2002. *Wind Turbine Design: with Emphasis on Darrieus Concept*. Presses inter Polytechnique, p. 472.
- Rainbird, J.M., Bianchini, A., Balduzzi, F., Peiró, J., Graham, J.M.R., Ferrara, G., Ferrari, L., 2015. On the influence of virtual camber effect on airfoil polars for use in simulations of Darrieus wind turbines. *Energy Convers. Manag.* 106, 373–384. <https://doi.org/10.1016/j.enconman.2015.09.053>.
- Saini, G., Kumar, A., Saini, R.P., 2021. Assessment of hydrokinetic energy – a case study of eastern Yamuna canal. *Mater. Today Proc.* 46, 5223–5227. <https://doi.org/10.1016/j.matpr.2020.08.595>.
- Shives, M., Crawford, C., Grove, S., 2017. A tuned actuator cylinder approach for predicting cross-flow turbine performance with wake interaction and channel blockage effects. *Int. J. Mar. Energy* 18, 30–56. <https://doi.org/10.1016/j.ijome.2017.03.007>.
- Sørensen, J.N., Dag, K.O., Ramos-García, N., 2014. A new tip correction based on the decambering approach. *J. Phys. Conf. Ser.* 524, 012097. <https://doi.org/10.1088/1742-6596/524/1/012097>.
- Viterna, L.A., Janetzke, D.C., 1982. Theoretical and experimental power from large horizontal-Axis wind turbines. *NASA Tech. Memo.* <https://doi.org/10.2172/6763041>.
- Yadav, P.K., Kumar, A., Jaiswal, S., 2023. A critical review of technologies for harnessing the power from flowing water using a hydrokinetic turbine to fulfill the energy need. *Energy Rep.* 9, 2102–2117. <https://doi.org/10.1016/j.egy.2023.01.033>.
- Yosry, A.G., Alvarez, E.A., Valdés, R.E., Pandal, A., Marigorta, E.B., 2023. Experimental and multiphase modeling of small vertical-axis hydrokinetic turbine with free-surface variations. *Renew. Energy* 203, 788–801. <https://doi.org/10.1016/j.renene.2022.12.114>.
- Yuce, M.I., Muratoglu, A., 2015. Hydrokinetic energy conversion systems: a technology status review. *Renew. Sustain. Energy Rev.* 43, 72–82. <https://doi.org/10.1016/j.rser.2014.10.037>.

CONCLUSIONS

This thesis investigates the complex hydrodynamics of Darrieus hydrokinetic turbines (HKTs) with a focus on understanding and improving the modelling of their performance within arrays and in realistic hydraulic environments. To achieve this, a multi-fidelity approach was employed, encompassing high-fidelity blade-resolved simulations, medium-fidelity Actuator Line Method (ALM) simulations, and a more computationally efficient Actuator Cylinder (AC) model. These numerical tools were validated and utilised to explore the underlying mechanisms of power augmentation within turbine arrays, to set design consideration for HKTs arrays, to assess the accuracy of different modelling approaches, and to simulate the two-way interactions between the turbine array and the surrounding flow, including the free water surface. Key findings from this thesis are summarised below.

Physics of Darrieus turbine in arrays

Through two-dimensional blade-resolved uRANS simulations, various in-line Darrieus turbine array configurations were investigated to understand the fundamental mechanisms responsible for performance enhancement compared to a stand-alone turbine. The findings from this study have been disseminated in two journal papers, referenced here as [P1, P2], with the following key conclusions:

- Closely spaced Darrieus turbines modify inflow patterns by suppressing streamtube expansion and straightening the inflow. This increases the angle of attack at azimuth positions adjacent to neighbouring turbines, leading to significant lift force enhancement at higher tip-speed ratios. This mechanism drives power augmentation within turbine arrays.
- The rotational arrangement of turbines within an array plays a critical role in determining overall efficiency. Configurations with counter-rotating leeward turbines generally achieve the highest efficiency gains.

ALM tuning in open-field conditions

Using two-dimensional uRANS within an assumed infinite domain, the ALM has been developed as an effective medium-fidelity tool for simulating the performance of Darrieus turbines. This included an assessment of the key factors influencing ALM accuracy, as well as an evaluation of its capability to capture power augmentation in turbine arrays. The findings from this study were published in [P3, P4], with the following key conclusions:

- Accurate representation of unsteady hydrodynamic forces is crucial for the ALM. While kernel shape has a minor impact on accuracy, the modelling of unsteady forces, particularly dynamic stall, is the most significant factor affecting the performance prediction.
- The ALM effectively captures instantaneous loads, local flow kinematics, and wake behaviour in closely spaced turbines. It provides significant computational advantages over blade-resolved simulations, making it a practical and efficient tool for realistic applications.

Modelling and validation of Darrieus HKTs under realistic conditions

Two different models were developed to simulate arrays of Darrieus HKTs while accounting for the free water surface and channel boundaries: ALM-CFD-VOF and AC-CFD-VOF. These models were validated against experimental measurements provided by the industrial partner, HE-PowerGreen, including the flow field, turbine array performance, and upstream water rise. The findings from this study were published in [P5, P6], with the key conclusions as follows:

- Incorporating channel geometries and free water surface interactions is crucial for accurately predicting turbine performance in real-world hydraulic environments. Such considerations in the design process enable the capture of complex two-way interactions between turbines and the channel, which are essential for assessing power coefficients and hydraulic impacts. These interactions significantly influence the design guidelines for HKTs.
- The AC model provides a computationally efficient tool for analysing turbines in arrays under realistic conditions but lacks the ability to capture detailed interactions between closely spaced turbines and the effects of channel blockage. In contrast, the ALM offers superior accuracy in representing these complex interactions, albeit at a higher computational cost..

These findings make a significant contribution to advancing the understanding of Darrieus HKT hydrodynamics and provide valuable tools for the design and optimisation of HKT arrays. The results emphasise the importance of considering complex interactions within turbine arrays and their surrounding environment, highlighting the need for accurate 3D modelling and the continuous improvement of numerical tools such as the ALM and AC. Therefore, it is recommended that these developed models be considered the standard for designing HKT arrays.

Building upon this work, future efforts should focus on

- **Enhancing the accuracy of the ALM and AC models:** This includes further development in the modelling of unsteady forces, particularly through the incorporation of advanced dynamic stall models such as the Beddoes-Leishman model. Additionally, future work should aim to improve the capability of the ALM to model tip effects.
- **Further analysis of the physics of Darrieus turbines in hydrokinetic applications:** Experimental investigations on the limits of power augmentation for Darrieus HKT should focus on the scalability of observed array behaviours and the influence of two-way interactions between the turbines and the channel.
- **Exploring alternative modelling techniques:** Investigating models based on Lifting Line Theory (LLT) or the Lattice Boltzmann Method (LBM) for the modelling of Darrieus HKT under realistic conditions and comparing their performance against the CFD-based models (ALM and AC) presented in this thesis.

REFERENCES

- [1] Kirke, B., 2020, "Hydrokinetic Turbines for Moderate Sized Rivers," *Energy Sustain Dev*, **58**, pp. 182–195. <https://doi.org/10.1016/j.esd.2020.08.003>.
- [2] Yuce, M. I., and Muratoglu, A., 2015, "Hydrokinetic Energy Conversion Systems: A Technology Status Review," *Renewable and Sustainable Energy Reviews*, **43**, pp. 72–82. <https://doi.org/10.1016/j.rser.2014.10.037>.
- [3] "Hydropower." [Online]. Available: https://energy.ec.europa.eu/topics/renewable-energy/hydropower_en. [Accessed: 18-Sep-2024].
- [4] Baring-Gould, E. I., Christol, C., LiVecchi, A., Kramer, S., and West, A., 2016, *A Review of the Environmental Impacts for Marine and Hydrokinetic Projects to Inform Regulatory Permitting: Summary Findings from the 2015 Workshop on Marine and Hydrokinetic Technologies, Washington, D.C.*, NREL/TP--5000-66688, 1263901. <https://doi.org/10.2172/1263901>.
- [5] Holanda, P. da S., Blanco, C. J. C., Mesquita, A. L. A., Brasil Junior, A. C. P., de Figueiredo, N. M., Macêdo, E. N., and Secretan, Y., 2017, "Assessment of Hydrokinetic Energy Resources Downstream of Hydropower Plants," *Renewable Energy*, **101**, pp. 1203–1214. <https://doi.org/10.1016/j.renene.2016.10.011>.
- [6] Montoya Ramírez, R. D., Cuervo, F. I., and Monsalve Rico, C. A., 2016, "Technical and Financial Valuation of Hydrokinetic Power in the Discharge Channels of Large Hydropower Plants in Colombia: A Case Study," *Renewable Energy*, **99**, pp. 136–147. <https://doi.org/10.1016/j.renene.2016.06.047>.
- [7] Saini, G., Kumar, A., and Saini, R. P., 2021, "Assessment of Hydrokinetic Energy – A Case Study of Eastern Yamuna Canal," *Materials Today: Proceedings*, **46**, pp. 5223–5227. <https://doi.org/10.1016/j.matpr.2020.08.595>.
- [8] Punys, P., Adamonyte, I., Kvaraciejus, A., Martinaitis, E., Vyciene, G., and Kasiulis, E., 2015, "Riverine Hydrokinetic Resource Assessment. A Case Study of a Lowland River in Lithuania," *Renewable and Sustainable Energy Reviews*, **50**(C), pp. 643–652.
- [9] Kusakana, K., and Vermaak, H. J., 2013, "Hydrokinetic Power Generation for Rural Electricity Supply: Case of South Africa," *Renewable Energy*, **55**, pp. 467–473. <https://doi.org/10.1016/j.renene.2012.12.051>.
- [10] Miller, V. B., Ramde, E. W., Gradoville, R. T., and Schaefer, L. A., 2011, "Hydrokinetic Power for Energy Access in Rural Ghana," *Renewable Energy*, **36**(2), pp. 671–675. <https://doi.org/10.1016/j.renene.2010.08.014>.
- [11] Botto, A., Claps, P., Ganora, D., and Laio, F., "Regional-Scale Assessment of Energy Potential from Hydrokinetic Turbines Used in Irrigation Channels."
- [12] Khan, M. J., Bhuyan, G., Iqbal, M. T., and Quaicoe, J. E., 2009, "Hydrokinetic Energy Conversion Systems and Assessment of Horizontal and Vertical Axis Turbines for River and Tidal Applications: A Technology Status Review," *Applied Energy*, **86**(10), pp. 1823–1835. <https://doi.org/10.1016/j.apenergy.2009.02.017>.
- [13] Runge, S., Stoesser, T., Morris, E., and White, M., 2018, "Technology Readiness of a Vertical-Axis Hydro-Kinetic Turbine," *Journal of Power and Energy Engineering*, **6**(8), pp. 63–85. <https://doi.org/10.4236/jpee.2018.68004>.

- [14] Dabiri, J. O., 2011, "Potential Order-of-Magnitude Enhancement of Wind Farm Power Density via Counter-Rotating Vertical-Axis Wind Turbine Arrays," *Journal of Renewable and Sustainable Energy*, **3**(4), p. 043104. <https://doi.org/10.1063/1.3608170>.
- [15] Ahmadi-Baloutaki, M., Cariveau, R., and Ting, D. S.-K., 2016, "A Wind Tunnel Study on the Aerodynamic Interaction of Vertical Axis Wind Turbines in Array Configurations," *Renewable Energy*, **96**, pp. 904–913. <https://doi.org/10.1016/j.renene.2016.05.060>.
- [16] Sahebzadeh, S., Rezaeiha, A., and Montazeri, H., 2020, "Towards Optimal Layout Design of Vertical-Axis Wind-Turbine Farms: Double Rotor Arrangements," *Energy Conversion and Management*, **226**, p. 113527. <https://doi.org/10.1016/j.enconman.2020.113527>.
- [17] Kinzel, M., Mulligan, Q., and Dabiri, J. O., 2012, "Energy Exchange in an Array of Vertical-Axis Wind Turbines," *Journal of Turbulence*, **13**, p. N38. <https://doi.org/10.1080/14685248.2012.712698>.
- [18] Vergaerde, A., De Troyer, T., Standaert, L., Kluczevska-Bordier, J., Pitance, D., Immas, A., Silvert, F., and Runacres, M. C., 2020, "Experimental Validation of the Power Enhancement of a Pair of Vertical-Axis Wind Turbines," *Renewable Energy*, **146**, pp. 181–187. <https://doi.org/10.1016/j.renene.2019.06.115>.
- [19] Li, S., Li, Y., Yang, C., Wang, Q., Zhao, B., Li, D., Zhao, R., Ren, T., Zheng, X., Gao, Z., and Xu, W., 2021, "Experimental Investigation of Solidity and Other Characteristics on Dual Vertical Axis Wind Turbines in an Urban Environment," *Energy Conversion and Management*, **229**, p. 113689. <https://doi.org/10.1016/j.enconman.2020.113689>.
- [20] Müller, S., Muhawenimana, V., Wilson, C. A. M. E., and Ouro, P., 2021, "Experimental Investigation of the Wake Characteristics behind Twin Vertical Axis Turbines," *Energy Conversion and Management*, **247**, p. 114768. <https://doi.org/10.1016/j.enconman.2021.114768>.
- [21] Talamalek, A., Runacres, M. C., and De Troyer, T., 2024, "Experimental Investigation of the Wake Replenishment Mechanisms of Paired Counter-Rotating Vertical-Axis Wind Turbines," *Journal of Wind Engineering and Industrial Aerodynamics*, **252**, p. 105830. <https://doi.org/10.1016/j.jweia.2024.105830>.
- [22] Jodai, Y., Tokuda, H., and Hara, Y., 2024, "Experiments on Interaction between Six Vertical-Axis Wind Turbines in Pairs or Trios," *J. Phys.: Conf. Ser.*, **2767**(7), p. 072003. <https://doi.org/10.1088/1742-6596/2767/7/072003>.
- [23] Sahebzadeh, S., Rezaeiha, A., and Montazeri, H., 2022, "Vertical-Axis Wind-Turbine Farm Design: Impact of Rotor Setting and Relative Arrangement on Aerodynamic Performance of Double Rotor Arrays," *Energy Reports*, **8**, pp. 5793–5819. <https://doi.org/10.1016/j.egyr.2022.04.030>.
- [24] Mohamed, O. S., Ibrahim, A., and El Baz, A. M. R., 2019, "CFD Investigation of the Multiple Rotors Darrieus Type Turbine Performance," *American Society of Mechanical Engineers Digital Collection*. <https://doi.org/10.1115/GT2019-91491>.
- [25] Zheng, H.-D., Zheng, X. Y., and Zhao, S. X., 2020, "Arrangement of Clustered Straight-Bladed Wind Turbines," *Energy*, **200**, p. 117563. <https://doi.org/10.1016/j.energy.2020.117563>.
- [26] Guilbot, M., Barre, S., Balarac, G., Bonamy, C., and Guillaud, N., 2020, "A Numerical Study of Vertical Axis Wind Turbine Performances in Twin-Rotor Configurations," *J. Phys.: Conf. Ser.*, **1618**(5), p. 052012. <https://doi.org/10.1088/1742-6596/1618/5/052012>.
- [27] Peng, H. Y., Han, Z. D., Liu, H. J., Lin, K., and Lam, H. F., 2020, "Assessment and Optimization of the Power Performance of Twin Vertical Axis Wind Turbines via Numerical Simulations," *Renewable Energy*, **147**, pp. 43–54. <https://doi.org/10.1016/j.renene.2019.08.124>.

- [28] Miller, M. A., Duvvuri, S., and Hultmark, M., 2021, "Solidity Effects on the Performance of Vertical-Axis Wind Turbines," *Flow*, **1**, p. E9. <https://doi.org/10.1017/flo.2021.9>.
- [29] Zanforlin, S., and Nishino, T., 2016, "Fluid Dynamic Mechanisms of Enhanced Power Generation by Closely Spaced Vertical Axis Wind Turbines," *Renewable Energy*, **99**, pp. 1213–1226. <https://doi.org/10.1016/j.renene.2016.08.015>.
- [30] Alexander, A. S., and Santhanakrishnan, A., 2020, "Mechanisms of Power Augmentation in Two Side-by-Side Vertical Axis Wind Turbines," *Renewable Energy*, **148**, pp. 600–610. <https://doi.org/10.1016/j.renene.2019.10.149>.
- [31] Zhao, P., Jiang, Y., Liu, S., Stoesser, T., Zou, L., and Wang, K., 2021, "Investigation of Fundamental Mechanism Leading to the Performance Improvement of Vertical Axis Wind Turbines by Deflector," *Energy Conversion and Management*, **247**, p. 114680. <https://doi.org/10.1016/j.enconman.2021.114680>.
- [32] Parneix, N., Fuchs, R., Immas, A., and Silvert, F., 2016, "EFFICIENCY IMPROVEMENT OF VERTICAL-AXIS WIND TURBINES WITH COUNTER-ROTATING LAY-OUT." [Online]. Available: <https://www.semanticscholar.org/paper/EFFICIENCY-IMPROVEMENT-OF-VERTICAL-AXIS-WIND-WITH-Parneix-Fuchs/f63f4a0af88b01151a2793add446701f92e18e89>. [Accessed: 15-Mar-2023].
- [33] Cacciali, L., Battisti, L., and Dell'Anna, S., 2023, "Multi-Array Design for Hydrokinetic Turbines in Hydropower Canals," *Energies*, **16**(5), p. 2279. <https://doi.org/10.3390/en16052279>.
- [34] Whittlesey, R. W., Liska, S., and Dabiri, J. O., 2010, "Fish Schooling as a Basis for Vertical Axis Wind Turbine Farm Design*," *Bioinspir. Biomim.*, **5**(3), p. 035005. <https://doi.org/10.1088/1748-3182/5/3/035005>.
- [35] Lapin, E. E., 1975, *Theoretical Performance of Vertical Axis Wind Turbines*. [Online]. Available: <https://ui.adsabs.harvard.edu/abs/1975asme.meetQ....L>. [Accessed: 04-Dec-2024].
- [36] Paraschivoiu, Ion, 1982, "Aerodynamic Loads and Performance of the Darrieus Rotor," *Journal of Energy*, **6**(6), pp. 406–412. <https://doi.org/10.2514/3.62621>.
- [37] Paraschivoiu, I., and Delclaux, F., 1983, "Double Multiple Streamtube Model with Recent Improvements (for Predicting Aerodynamic Loads and Performance of Darrieus Vertical Axis Wind Turbines)," *Journal of Energy*, **7**(3), pp. 250–255. <https://doi.org/10.2514/3.48077>.
- [38] Cacciali, L., Battisti, L., and Dell'Anna, S., 2022, "Free Surface Double Actuator Disc Theory and Double Multiple Streamtube Model for In-Stream Darrieus Hydrokinetic Turbines," *Ocean Engineering*, **260**, p. 112017. <https://doi.org/10.1016/j.oceaneng.2022.112017>.
- [39] Cacciali, L., Battisti, L., and Dell'Anna, S., 2023, "Backwater Assessment for the Energy Conversion through Hydrokinetic Turbines in Subcritical Prismatic Canals," *Ocean Engineering*, **267**, p. 113246. <https://doi.org/10.1016/j.oceaneng.2022.113246>.
- [40] Wilson, R., and Mckie, W., 1980, "A Comparison of Aerodynamic Analyses for the Darrieus Rotor," *Wind Energy Conference*, American Institute of Aeronautics and Astronautics. <https://doi.org/10.2514/6.1980-605>.
- [41] Hansen, M., 2015, *Aerodynamics of Wind Turbines*, Routledge, London. <https://doi.org/10.4324/9781315769981>.
- [42] Ayati, A. A., Steiros, K., Miller, M. A., Duvvuri, S., and Hultmark, M., 2019, "A Double-Multiple Streamtube Model for Vertical Axis Wind Turbines of Arbitrary Rotor Loading," *Wind Energy Science*, **4**(4), pp. 653–662. <https://doi.org/10.5194/wes-4-653-2019>.
- [43] Madsen, H., 1982, "The Actuator Cylinder - A Flow Model for Vertical Axis Wind Turbines." <https://doi.org/10.13140/RG.2.1.2512.3040>.

- [44] Madsen, H. A., Paulsen, U. S., and Vitae, L., 2014, "Analysis of VAWT Aerodynamics and Design Using the Actuator Cylinder Flow Model," *J. Phys.: Conf. Ser.*, **555**(1), p. 012065. <https://doi.org/10.1088/1742-6596/555/1/012065>.
- [45] Shamsoddin, S., and Porté-Agel, F., 2014, "Large Eddy Simulation of Vertical Axis Wind Turbine Wakes," *Energies*, **7**(2), pp. 890–912. <https://doi.org/10.3390/en7020890>.
- [46] Ning, A., 2016, "Actuator Cylinder Theory for Multiple Vertical Axis Wind Turbines," *Wind Energy Science*, **1**(2), pp. 327–340. <https://doi.org/10.5194/wes-1-327-2016>.
- [47] Shives, M., Crawford, C., and Grovum, S., 2017, "A Tuned Actuator Cylinder Approach for Predicting Cross-Flow Turbine Performance with Wake Interaction and Channel Blockage Effects," *International Journal of Marine Energy*, **18**, pp. 30–56. <https://doi.org/10.1016/j.ijome.2017.03.007>.
- [48] Murray, J., and Barone, M., 2011, "The Development of CACTUS, a Wind and Marine Turbine Performance Simulation Code," *49th AIAA Aerospace Sciences Meeting Including the New Horizons Forum and Aerospace Exposition*, American Institute of Aeronautics and Astronautics, Orlando, Florida. <https://doi.org/10.2514/6.2011-147>.
- [49] Marten, D., Pechlivanoglou, G., Nayeri, C., and Paschereit, C. O., 2016, "Nonlinear Lifting Line Theory Applied To Vertical Axis Wind Turbines: Development of a Practical Design Tool," *16th International Symposium on Transport Phenomena and Dynamics of Rotating Machinery*, Honolulu, United States. [Online]. Available: <https://hal.science/hal-01887470>. [Accessed: 09-Dec-2024].
- [50] Shaler, K., Branlard, E., and Platt, A., 2020, *OLAF User's Guide and Theory Manual*, NREL/TP-5000-75959, 1659853, MainId:6799. <https://doi.org/10.2172/1659853>.
- [51] Branlard, E., Brownstein, I., Strom, B., Jonkman, J., Dana, S., and Baring-Gould, E. I., 2022, "A Multipurpose Lifting-Line Flow Solver for Arbitrary Wind Energy Concepts," *Wind Energ. Sci.*, **7**(2), pp. 455–467. <https://doi.org/10.5194/wes-7-455-2022>.
- [52] Tavernier, D. D., Ferreira, C., Li, A., Paulsen, U. S., and Madsen, H. A., 2018, "Towards the Understanding of Vertical-Axis Wind Turbines in Double-Rotor Configuration.," *J. Phys.: Conf. Ser.*, **1037**(2), p. 022015. <https://doi.org/10.1088/1742-6596/1037/2/022015>.
- [53] So/rensen, J. N., and Shen, W. Z., 2002, "Numerical Modeling of Wind Turbine Wakes," *Journal of Fluids Engineering*, **124**(2), pp. 393–399. <https://doi.org/10.1115/1.1471361>.
- [54] Bai, C.-J., and Wang, W.-C., 2016, "Review of Computational and Experimental Approaches to Analysis of Aerodynamic Performance in Horizontal-Axis Wind Turbines (HAWTs)," *Renewable and Sustainable Energy Reviews*, **63**, pp. 506–519. <https://doi.org/10.1016/j.rser.2016.05.078>.
- [55] Shamsoddin, S., and Porté-Agel, F., 2016, "A Large-Eddy Simulation Study of Vertical Axis Wind Turbine Wakes in the Atmospheric Boundary Layer," *Energies*, **9**(5), p. 366. <https://doi.org/10.3390/en9050366>.
- [56] Abkar, M., and Dabiri, J. O., 2017, "Self-Similarity and Flow Characteristics of Vertical-Axis Wind Turbine Wakes: An LES Study," *Journal of Turbulence*, **18**(4), pp. 373–389. <https://doi.org/10.1080/14685248.2017.1284327>.
- [57] Mendoza, V., Bachant, P., Ferreira, C., and Goude, A., 2019, "Near-Wake Flow Simulation of a Vertical Axis Turbine Using an Actuator Line Model," *Wind Energy*, **22**(2), pp. 171–188. <https://doi.org/10.1002/we.2277>.
- [58] Melani, P. F., Balduzzi, F., Ferrara, G., and Bianchini, A., 2021, "Tailoring the Actuator Line Theory to the Simulation of Vertical-Axis Wind Turbines," *Energy Conversion and Management*, **243**, p. 114422. <https://doi.org/10.1016/j.enconman.2021.114422>.

- [59] Jost, E., Klein, L., Leipprand, H., Lutz, T., and Krämer, E., 2018, "Extracting the Angle of Attack on Rotor Blades from CFD Simulations," *Wind Energy*, **21**(10), pp. 807–822. <https://doi.org/10.1002/we.2196>.
- [60] Bianchini, A., Balduzzi, F., Rainbird, J. M., Peiro, J., Graham, J. M. R., Ferrara, G., and Ferrari, L., 2015, "An Experimental and Numerical Assessment of Airfoil Polars for Use in Darrieus Wind Turbines: Part 1 — Flow Curvature Effects," American Society of Mechanical Engineers Digital Collection. <https://doi.org/10.1115/GT2015-42284>.
- [61] Bianchini, A., Balduzzi, F., Ferrara, G., and Ferrari, L., 2016, "Aerodynamics of Darrieus Wind Turbines Airfoils: The Impact of Pitching Moment," *Journal of Engineering for Gas Turbines and Power*, **139**(4). <https://doi.org/10.1115/1.4034940>.
- [62] Bianchini, A., Balduzzi, F., Ferrara, G., and Ferrari, L., 2016, "Critical Analysis of Dynamic Stall Models in Low-Order Simulation Models For Vertical-Axis Wind Turbines," *Energy Procedia*, **101**, pp. 488–495. <https://doi.org/10.1016/j.egypro.2016.11.062>.
- [63] Balduzzi, F., Bianchini, A., Maleci, R., Ferrara, G., and Ferrari, L., 2016, "Critical Issues in the CFD Simulation of Darrieus Wind Turbines," *Renewable Energy*, **85**, pp. 419–435. <https://doi.org/10.1016/j.renene.2015.06.048>.
- [64] Mohamed, O. S., Melani, P. F., Balduzzi, F., Ferrara, G., and Bianchini, A., 2023, "An Insight on the Physical Mechanisms Responsible of Power Augmentation in a Pair of Counter-Rotating Darrieus Turbines," *Energy Conversion and Management*, **284**, p. 116991. <https://doi.org/10.1016/j.enconman.2023.116991>.
- [65] Melani, P. F., Balduzzi, F., Ferrara, G., and Bianchini, A., 2020, "How to Extract the Angle Attack on Airfoils in Cycloidal Motion from a Flow Field Solved with Computational Fluid Dynamics? Development and Verification of a Robust Computational Procedure," *Energy Conversion and Management*, **223**, p. 113284. <https://doi.org/10.1016/j.enconman.2020.113284>.
- [66] Mohamed, O. S., Elbaz, A. M. R., and Bianchini, A., 2021, "A Better Insight on Physics Involved in the Self-Starting of a Straight-Blade Darrieus Wind Turbine by Means of Two-Dimensional Computational Fluid Dynamics," *Journal of Wind Engineering and Industrial Aerodynamics*, **218**, p. 104793. <https://doi.org/10.1016/j.jweia.2021.104793>.
- [67] Franchina, N., Persico, G., and Savini, M., 2019, "2D-3D Computations of a Vertical Axis Wind Turbine Flow Field: Modeling Issues and Physical Interpretations," *Renewable Energy*, **136**, pp. 1170–1189. <https://doi.org/10.1016/j.renene.2018.09.086>.
- [68] He, J., Jin, X., Xie, S., Cao, L., Wang, Y., Lin, Y., and Wang, N., 2020, "CFD Modeling of Varying Complexity for Aerodynamic Analysis of H-Vertical Axis Wind Turbines," *Renewable Energy*, **145**, pp. 2658–2670. <https://doi.org/10.1016/j.renene.2019.07.132>.
- [69] Ferreira, C. S., Madsen, H. A., Barone, M., Roscher, B., Deglaire, P., and Arduin, I., 2014, "Comparison of Aerodynamic Models for Vertical Axis Wind Turbines," *J. Phys.: Conf. Ser.*, **524**, p. 012125. <https://doi.org/10.1088/1742-6596/524/1/012125>.
- [70] Cheng, Z., Madsen, H. A., Gao, Z., and Moan, T., 2016, "Aerodynamic Modeling of Floating Vertical Axis Wind Turbines Using the Actuator Cylinder Flow Method," *Energy Procedia*, **94**, pp. 531–543. <https://doi.org/10.1016/j.egypro.2016.09.232>.
- [71] Delafin, P.-L., Nishino, T., Kolios, A., and Wang, L., 2017, "Comparison of Low-Order Aerodynamic Models and RANS CFD for Full Scale 3D Vertical Axis Wind Turbines," *Renewable Energy*, **109**, pp. 564–575. <https://doi.org/10.1016/j.renene.2017.03.065>.
- [72] Tavernier, D. D., Sakib, M., Griffith, T., Pirrung, G., Paulsen, U., Madsen, H., Keijer, W., and Ferreira, C., 2020, "Comparison of 3D Aerodynamic Models for Vertical-Axis Wind Turbines:

- H-Rotor and Φ -Rotor," J. Phys.: Conf. Ser., **1618**(5), p. 052041. <https://doi.org/10.1088/1742-6596/1618/5/052041>.
- [73] Giri Ajay, A., Morgan, L., Wu, Y., Bretos, D., Cascales, A., Pires, O., and Ferreira, C., 2024, "Aerodynamic Model Comparison for an X-Shaped Vertical-Axis Wind Turbine," Wind Energy Science, **9**(2), pp. 453–470. <https://doi.org/10.5194/wes-9-453-2024>.
- [74] Garrett, C., and Cummins, P., 2007, "The Efficiency of a Turbine in a Tidal Channel," Journal of Fluid Mechanics, **588**, pp. 243–251. <https://doi.org/10.1017/S0022112007007781>.
- [75] Nishino, T., and Willden, R. H. J., 2012, "The Efficiency of an Array of Tidal Turbines Partially Blocking a Wide Channel," J. Fluid Mech., **708**, pp. 596–606. <https://doi.org/10.1017/jfm.2012.349>.
- [76] Malki, R., Masters, I., Williams, A. J., and Nick Croft, T., 2014, "Planning Tidal Stream Turbine Array Layouts Using a Coupled Blade Element Momentum – Computational Fluid Dynamics Model," Renewable Energy, **63**, pp. 46–54. <https://doi.org/10.1016/j.renene.2013.08.039>.
- [77] Kirke, B., 2019, "Hydrokinetic and Ultra-Low Head Turbines in Rivers: A Reality Check," Energy for Sustainable Development, **52**, pp. 1–10. <https://doi.org/10.1016/j.esd.2019.06.002>.
- [78] Kinsey, T., and Dumas, G., 2017, "Impact of Channel Blockage on the Performance of Axial and Cross-Flow Hydrokinetic Turbines," Renewable Energy, **103**, pp. 239–254. <https://doi.org/10.1016/j.renene.2016.11.021>.
- [79] Gauvin-Tremblay, O., and Dumas, G., 2020, "Two-Way Interaction between River and Deployed Cross-Flow Hydrokinetic Turbines," Journal of Renewable and Sustainable Energy, **12**(3), p. 034501. <https://doi.org/10.1063/5.0004492>.
- [80] Myers, L., and Bahaj, A. S., 2007, "Wake Studies of a 1/30th Scale Horizontal Axis Marine Current Turbine," Ocean Engineering, **34**(5), pp. 758–762. <https://doi.org/10.1016/j.oceaneng.2006.04.013>.
- [81] Chaulagain, R. K., Poudel, L., and Maharjan, S., 2023, "A Review on Non-Conventional Hydropower Turbines and Their Selection for Ultra-Low-Head Applications," Heliyon, **9**(7), p. e17753. <https://doi.org/10.1016/j.heliyon.2023.e17753>.
- [82] Yosry, A. G., Álvarez, E. Á., Valdés, R. E., Pandal, A., and Marigorta, E. B., 2023, "Experimental and Multiphase Modeling of Small Vertical-Axis Hydrokinetic Turbine with Free-Surface Variations," Renewable Energy, **203**, pp. 788–801. <https://doi.org/10.1016/j.renene.2022.12.114>.
- [83] Hirt, C. W., and Nichols, B. D., 1981, "Volume of Fluid (VOF) Method for the Dynamics of Free Boundaries," Journal of Computational Physics, **39**(1), pp. 201–225. [https://doi.org/10.1016/0021-9991\(81\)90145-5](https://doi.org/10.1016/0021-9991(81)90145-5).
- [84] "ANSYS FLUENT 12.0 Theory Guide." [Online]. Available: https://www.afs.enea.it/project/neptunius/docs/fluent/html/th/main_pre.htm. [Accessed: 20-Nov-2023].
- [85] Bai, X., Avital, E. J., Munjiza, A., and Williams, J. J. R., 2014, "Numerical Simulation of a Marine Current Turbine in Free Surface Flow," Renewable Energy, **63**, pp. 715–723. <https://doi.org/10.1016/j.renene.2013.09.042>.
- [86] Kolekar, N., and Banerjee, A., 2015, "Performance Characterization and Placement of a Marine Hydrokinetic Turbine in a Tidal Channel under Boundary Proximity and Blockage Effects," Applied Energy, **148**, pp. 121–133. <https://doi.org/10.1016/j.apenergy.2015.03.052>.

- [87] Nishi, Y., Sato, G., Shiohara, D., Inagaki, T., and Kikuchi, N., 2017, "Performance Characteristics of Axial Flow Hydraulic Turbine with a Collection Device in Free Surface Flow Field," *Renewable Energy*, **112**, pp. 53–62. <https://doi.org/10.1016/j.renene.2017.04.047>.
- [88] Benavides-Morán, A., Rodríguez-Jaime, L., and Laín, S., 2022, "Numerical Investigation of the Performance, Hydrodynamics, and Free-Surface Effects in Unsteady Flow of a Horizontal Axis Hydrokinetic Turbine," *Processes*, **10**(1), p. 69. <https://doi.org/10.3390/pr10010069>.
- [89] Apsley, D., and Hu, W., 2003, "CFD Simulation of Two- and Three-Dimensional Free-Surface Flow," *International Journal for Numerical Methods in Fluids*, **42**(5), pp. 465–491. <https://doi.org/10.1002/fld.523>.
- [90] Apsley, D. D., Stallard, T., and Stansby, P. K., 2018, "Actuator-Line CFD Modelling of Tidal-Stream Turbines in Arrays," *J. Ocean Eng. Mar. Energy*, **4**(4), pp. 259–271. <https://doi.org/10.1007/s40722-018-0120-3>.
- [91] Melani, P. F., Balduzzi, F., Ferrara, G., and Bianchini, A., 2021, "Development of a Desmodromic Variable Pitch System for Hydrokinetic Turbines," *Energy Conversion and Management*, **250**, p. 114890. <https://doi.org/10.1016/j.enconman.2021.114890>.
- [92] Hara, Y., Jodai, Y., Okinaga, T., and Furukawa, M., 2021, "Numerical Analysis of the Dynamic Interaction between Two Closely Spaced Vertical-Axis Wind Turbines," *Energies*, **14**(8), p. 2286. <https://doi.org/10.3390/en14082286>.
- [93] Nazari, S., Zamani, M., and Moshizi, S. A., 2018, "Comparison between Two-Dimensional and Three-Dimensional Computational Fluid Dynamics Techniques for Two Straight-Bladed Vertical-Axis Wind Turbines in Inline Arrangement," *Wind Engineering*, **42**(6), pp. 647–664. <https://doi.org/10.1177/0309524X18780384>.
- [94] Martínez-Tossas, L. A., Churchfield, M. J., and Meneveau, C., 2016, "A Highly Resolved Large-Eddy Simulation of a Wind Turbine Using an Actuator Line Model with Optimal Body Force Projection," *J. Phys.: Conf. Ser.*, **753**(8), p. 082014. <https://doi.org/10.1088/1742-6596/753/8/082014>.
- [95] Churchfield, M. J., Schreck, S. J., Martinez, L. A., Meneveau, C., and Spalart, P. R., "An Advanced Actuator Line Method for Wind Energy Applications and Beyond," *35th Wind Energy Symposium*, American Institute of Aeronautics and Astronautics. <https://doi.org/10.2514/6.2017-1998>.
- [96] Cormier, M., Weihing, P., and Lutz, T., 2021, "Evaluation of the Effects of Actuator Line Force Smearing on Wind Turbines Near-Wake Development," *J. Phys.: Conf. Ser.*, **1934**(1), p. 012013. <https://doi.org/10.1088/1742-6596/1934/1/012013>.
- [97] "Il Canale Biffis - Claudio Malini - Libro - Sometti - Acque e bonifiche | IBS." [Online]. Available: <https://www.ibs.it/canale-biffis-libro-claudio-malini/e/9788874956760>. [Accessed: 22-Feb-2024].



UNIONE EUROPEA
Fondo Sociale Europeo



*Ministero dell'Università
e della Ricerca*



La borsa di dottorato è stata cofinanziata con risorse del
Programma Operativo Nazionale Ricerca e Innovazione 2014-2020, risorse FSE REACT-EU
Azione IV.4 “Dottorati e contratti di ricerca su tematiche dell’innovazione”
e Azione IV.5 “Dottorati su tematiche Green

