



UNIVERSITÀ
DEGLI STUDI
FIRENZE

FLORE

Repository istituzionale dell'Università degli Studi di Firenze

Analytical and computational fluid dynamics models of Wells turbines for oscillating water column systems

Questa è la Versione finale referata (Post print/Accepted manuscript) della seguente pubblicazione:

Original Citation:

Analytical and computational fluid dynamics models of Wells turbines for oscillating water column systems / Lorenzo Ciappi, L.C.. - In: JOURNAL OF ENERGY RESOURCES TECHNOLOGY. - ISSN 0195-0738. - ELETTRONICO. - 144:(2022), pp. 1-13. [10.1115/1.4052216]

Availability:

The webpage <https://hdl.handle.net/2158/1245360> of the repository was last updated on 2025-01-21T14:11:15Z

Published version:

DOI: 10.1115/1.4052216

Terms of use:

Open Access

La pubblicazione è resa disponibile sotto le norme e i termini della licenza di deposito, secondo quanto stabilito dalla Policy per l'accesso aperto dell'Università degli Studi di Firenze (<https://www.sba.unifi.it/upload/policy-oa-2016-1.pdf>)

Publisher copyright claim:

Conformità alle politiche dell'editore / Compliance to publisher's policies

Questa versione della pubblicazione è conforme a quanto richiesto dalle politiche dell'editore in materia di copyright.

This version of the publication conforms to the publisher's copyright policies.

La data sopra indicata si riferisce all'ultimo aggiornamento della scheda del Repository FloRe - The above-mentioned date refers to the last update of the record in the Institutional Repository FloRe

(Article begins on next page)

Analytical and Computational Fluid Dynamics Models of Wells Turbines for Oscillating Water Column Systems

L. Ciappi¹

Department of Industrial Engineering,
University of Florence,
Firenze 50134, Italy
e-mail: lorenzo.ciappi@unifi.it

M. Stebel

Institute of Thermal Technology,
Silesian University of Technology,
Gliwice 44-100, Poland
e-mail: michal.stebel@polsl.pl

J. Smolka

Institute of Thermal Technology,
Silesian University of Technology,
Gliwice 44-100, Poland
e-mail: jacek.smolka@polsl.pl

L. Cappiotti

Department of Civil and Environmental
Engineering,
University of Florence,
Firenze 50139, Italy
e-mail: lorenzo.cappiotti@unifi.it

G. Manfrida

Department of Industrial Engineering,
University of Florence,
Firenze 50134, Italy
e-mail: giampaolo.manfrida@unifi.it

The sea is an important renewable energy resource for its extension and the power conveyed by waves, currents, tides, and thermal gradients. Amongst these physical phenomena, sea waves are the source with the highest energy density and may contribute to fulfilling the global increase of power demand. Despite the potential of sea waves, their harnessing is still a technological challenge. Oscillating water column systems operating with Wells turbines represent one of the most straightforward and reliable solutions for the optimal exploitation of this resource. An analytical model and computational fluid dynamics models were developed to evaluate the functioning of monoplane isolated Wells turbines. For the former modeling typology, a blade element momentum code relying on the actuator disk theory was applied, considering the rotor as a set of airfoils. For the latter modeling typology, a three-dimensional multi-block technique was implemented to create the computational domain with a fully mapped mesh composed of hexahedral elements. The employment of circumferential periodic boundary conditions allowed for the reduction of computational power and time. The models use Reynolds-averaged Navier-Stokes (RANS) or u-RANS schemes with a multiple reference frame approach or the u-RANS formulation with a sliding mesh approach. The achieved results were compared with analytical and experimental literature data for validation. All the developed models showed good agreement. The analytical model is suitable for a fast prediction of the turbine operation on a wide set of configurations during the first design stages, while the computational fluid dynamics (CFD) models are indicated for the further investigation of the selected configurations. [DOI: 10.1115/1.4052216]

Keywords: Wells turbine, analytical models, CFD models, OWC system, wave energy, alternative energy sources, renewable energy

1 Introduction

In recent years, energy demand has been continuously growing worldwide. Together with the possible exhaustion of fossil fuels and the level of pollutant and greenhouse emissions, the increasing energy request has raised the global efforts toward enhancing the harvesting of renewable resources. In this framework, the sea is one of the renewable energies with the highest potential. The physical phenomena that can be exploited are waves, currents, tides, and thermal gradients. Since sea waves combine the highest energy density and availability amongst the cited ones, wave energy conversion systems (WECs) are attractive for providing a contribution to the electric energy supply.

Wave harvesting has a remarkable perspective, as the wave power incident on the world coastline is estimated among 0.5 and 2.2 TW [1,2], which corresponds to more than 10% of the global power consumption [3]. Coupled with the abundant resource availability, an essential aspect of energy transformation from waves involves not impacting and not polluting technologies. Besides, WECs are prone to be coupled with photovoltaic plants and offshore wind turbines [4]. There are three main categories of wave energy conversion systems [5], classified on their working

principle: oscillating water column devices (OWCs), overtopping devices (OTDs), and oscillating body devices (OBDs). Different from other renewable energy fields, a predominant technology has not yet been identified, and power harnessing still represents a challenge [6]. The OWC devices are considered very promising due to the structural simplicity, reliability, and absence of mechanical parts moving under water [7].

The research on air turbines for OWC systems showed particular interest in Wells [8–10] and impulse turbines [11–13]. In the following, the attention is focused on analytical and numerical models of Wells turbines, which are the topic of the present study.

In 1984, Gato and Falcão presented a theoretical investigation of the aerodynamics of a monoplane isolated Wells turbine functioning with a reciprocating airflow. The flow crossing the air turbine and the main performance parameters were analyzed through the application of either the two-dimensional cascade theory or the actuator disk theory [9]. In 1985, Raghunathan et al. analytically and experimentally evaluated the effects of the principal geometric and aerodynamic parameters on the operation of monoplane isolated Wells turbines. Two analytical models relying on the two-dimensional cascade theory and the isolated airfoil data were developed. The experiments were performed on a rig with unidirectional airflow and a rig with oscillating airflow [14]. In 1991, Raghunathan et al. further investigated the effects of the main design variables on the turbine functioning and proposed techniques for improving its performance [15]. In 1995, Raghunathan defined a methodology for designing Wells turbines for wave power plants based on an iterative calculation of the geometric and aerodynamic parameters [16].

¹Corresponding author.

Contributed by the Advanced Energy Systems Division of ASME for publication in the JOURNAL OF ENERGY RESOURCES TECHNOLOGY. Manuscript received March 31, 2021; final manuscript received August 16, 2021; published online September 15, 2021. Assoc. Editor: Tatiana Morosuk.

The same author reviewed the analytical and experimental models for predicting the operation of monoplane and biplane Wells turbines [17].

In 2002, Brito-Melo et al. applied computational fluid dynamics (CFD) to investigate the impact on the aerodynamics of the design solutions of Wells turbines, considering the presence or absence of guide vanes and a bypass relief valve [18]. In 2003, Heikal applied a two-dimensional analysis relying on the radial equilibrium theory to determine the influence of the blade shape, rotor solidity, and Reynolds number on the functioning of turbomachines with and without guide vanes. The outcomes were compared with experimental data obtained with a unidirectional airflow rig [19]. In 2004, Torresi et al. performed CFD analyses to evaluate the blade tip leakage and the hysteretic mechanism under oscillating flow conditions for a low-solidity Wells turbine [20]. In 2008, the same authors described the steady three-dimensional flow field of a machine with high rotor solidity. A uniform tip clearance gap was analyzed, observing that a tip clearance ratio of around 0.01 increases turbine performance and reduces the impact of this parameter [21]. In 2010, Taha et al. carried out steady-state CFD analyses to determine the effects of different values of uniform tip clearance gap for a Wells turbine with blade profile NACA0020 under steady flow conditions [22]. The following year, the analyses were extended to non-uniform dimensions showing very slight positive effects on the overall turbine performance. The numerical results were compared with literature experiments in terms of input and torque coefficients and efficiency [23]. In 2011, Torresi et al. presented an actuator disk model of a high-solidity turbine and compared the predicted performance with the outcomes of the corresponding CFD model. Either the approaches were utilized to assess the operation of a monoplane isolated Wells turbine with a chord length linearly growing with the rotor radius [24]. Starzmann and Carolus numerically analyzed the impact of the hub-to-tip ratio and the rotor solidity on the aerodynamics and aeroacoustics, demonstrating that lowering these parameters enlarges the operating regime without stalling and reduces the noise emissions [25]. In 2014, Bassetti et al. developed a blade element momentum (BEM) model and applied numerical analyses to build a prototype of a wave energy converter for evaluating the aerodynamics and acoustics of Wells turbines [26].

The most recent literature indicates that the research on oscillating water column systems is moving toward the comprehensive study of the entire process of power conversion from sea waves to the electric wire [27–31]. An extensive literature review on the related design solutions and modeling approaches was proposed by some of the authors of this study [30].

The analysis of the existing literature reveals a lack in the comparison between analytical and numerical models for the evaluation of the aerodynamics of Wells turbines. In the present work, a BEM model and CFD models were developed, compared, and validated against analytical and experimental literature data, showing good agreement [26,32]. The models thoroughly consider the processes of energy transformation occurring in the secondary converter of an OWC system based on the pressure gradients alternatively made available by the chamber. A detailed analysis of the fluid dynamics and thermodynamics of the turbine was conducted with either the approaches alongside the assessment of the main performance parameters.

Furthermore, the analytical code can operate in stand-alone mode or be integrated into a wave-to-wire model to investigate the functioning of the entire system. Therefore, comparing the analytical model with more accurate models such as numerical and experimental ones is paramount to verify its accuracy in predictions.

2 Subject of the Study

The Wells turbine is an axial-flow turbine invented by A. A. Wells in 1976 and it is the most widespread turbomachine for energy conversion from sea waves [8]. The literature case

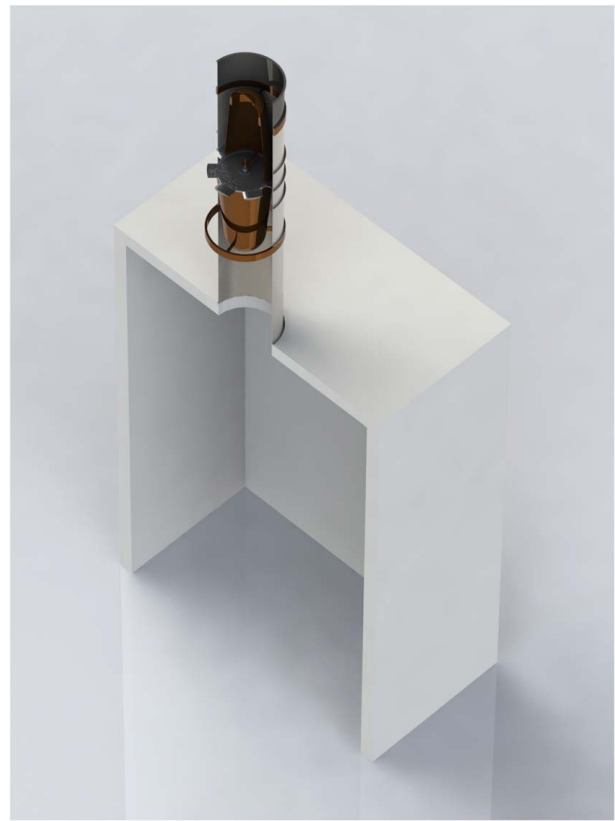


Fig. 1 Section of the rendering of an OWC system showing the chamber and the monoplane isolated Wells turbine

study is an isolated monoplane Wells turbine composed of a single rotor cascade with symmetrical blades staggered around the central hub, symmetrically positioned regarding the plane normal to the rotation axis and thus at 90 deg concerning the incoming flow (Fig. 1).

The main geometric specifications and operating conditions of the Wells turbine investigated are listed in Table 1 based on the reference literature data [26,32]. For the analyses performed by imposing a sinusoidal law as a function of time for the inlet velocity, a wave period of 7 s was selected to consider typical conditions of a moderate climate of the Mediterranean Sea [30,31].

3 Analytical Model

A blade element momentum model applying the actuator disk theory was developed for evaluating the aerodynamics of monoplane isolated Wells turbines. With this approach, a turbomachine

Table 1 Main geometric specifications and operating conditions of the Wells turbine

Parameter	Description
Turbine typology	Isolated monoplane
Blade profile	NACA0015
Blade number	7
Turbine casing diameter	0.50 m
Hub-to-tip ratio	0.75
Chord length	0.117 m
Rotational speed	3600 rpm
Inflow velocity amplitude	14.91 m/s
Outflow velocity amplitude	15.73 m/s
Airflow period	7 s

row is modeled as an infinitely thin disk simulating the former in terms of energy exchange with the flow through the imposition of mass and momentum conservation [33,34]. The flow is assumed to be steady, axisymmetric, and inviscid. Furthermore, the blade element momentum theory models each blade as a finite set of airfoils stacked along the blade span. Every couple of adjacent airfoils defines a blade element [29,31].

Since every blade element has its distinctive geometry and rotates with a specific tangential speed, undergoing particular flow conditions, a sensitivity analysis to the element number was performed. It was found that 50 blade elements are the optimal compromise between accuracy and computational time.

At the inlet and outlet of the domain, ideally positioned far upstream and far downstream of the actuator disk, the condition of radial equilibrium is assigned. The flow entering the domain of a monoplane isolated Wells turbine is assumed perfectly axial and uniform for either the inflow and outflow phases. Consequently, the axial component of the inlet absolute velocity is imposed by the continuity equation, and its circumferential component is null. The velocity triangles at the inlet section can be calculated for each blade element by varying the flow conditions and the rotor angular speed (Fig. 2).

The solution of the aerodynamics is based on the use of the lift and drag coefficients C_L and C_D , following the classical airfoil theory [29,31]. For this purpose, the experimental data of the Sandia tables measured on isolated profiles are utilized. In turbomachinery, the presence of the clearance determines the leakage of part of the airflow through the tip gap and thus a reduction of the power conversion. Therefore, the experimental drag coefficients are increased by adding induced drag coefficients to obtain equivalent drag coefficients C_{Dtc} modeling the clearance effect through Eq. (1). The induced drag coefficient of a generic blade element i depends on its lift coefficient, the tip clearance gap t , the aspect ratio AR , and the pitch τ computed at the blade tip [35]

$$C_{Dtc,i} = C_{D,i} + 0.7 \frac{C_{L,i} t}{AR \tau} \quad (1)$$

Afterward, the lift and equivalent drag coefficients are applied in Eqs. (2) and (3) to determine the lift and drag forces f_L and f_D developed by a flow impacting an infinitesimal blade element of chord

length l with relative velocity w

$$df_{L,i} = \frac{1}{2} \rho l_i C_{L,i} w_{1,i}^2 dr \quad (2)$$

$$df_{D,i} = \frac{1}{2} \rho l_i C_{Dtc,i} w_{1,i}^2 dr \quad (3)$$

where ρ is the air density, r is the turbine radius, and subscript i indicates the rotor inlet.

A trigonometric combination of the lift and drag forces is applied in Eqs. (4) and (5) for computing the axial and tangential components f_x and f_θ acting on every blade element

$$df_{x,i} = df_{L,i} \cos \beta_{1,i} + df_{D,i} \sin \beta_{1,i} = \frac{1}{2} \rho a_i l_i C_{x,i} w_{1,i}^2 dr \quad (4)$$

$$df_{\theta,i} = df_{L,i} \sin \beta_{1,i} - df_{D,i} \cos \beta_{1,i} = \frac{1}{2} \rho a_i l_i C_{\theta,i} w_{1,i}^2 dr \quad (5)$$

where β is the incidence angle of the flow on the blade, and C_x and C_θ are the axial and tangential force coefficients, respectively.

The midspan solidity of Wells turbines ranges between 0.4 and 0.6 to operate efficiently with self-starting mode. This implies a relevant mutual interaction among the blades of the cascade due to inviscid and viscous effects. Therefore, the use of a correction factor is required to consider this phenomenon. The cascade interference factor (CIF) of Eq. (6) based on the rotor solidity σ is utilized for this purpose [17]

$$\text{CIF} = \frac{1}{1 - \sigma_i^2} \quad (6)$$

The cascade interference factor is multiplied by C_x and C_θ through Eqs. (7) and (8) to modify the values of the axial and tangential force coefficients. The coefficients C_{xCIF} and $C_{\theta CIF}$ obtained are substituted into Eqs. (4) and (5) to the corresponding unperturbed values for computing the axial and tangential forces altered by the mutual blade interaction

$$C_{xCIF,i} = \text{CIF} C_{x,i} \quad (7)$$

$$C_{\theta CIF,i} = \text{CIF} C_{\theta,i} \quad (8)$$

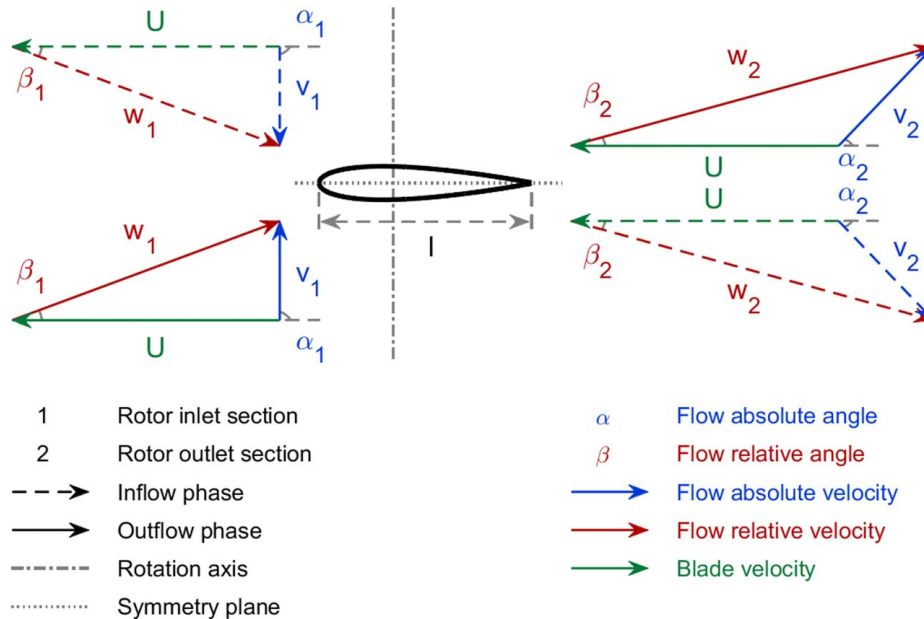


Fig. 2 Scheme of the blade-to-blade section and velocity triangles of a monoplane isolated Wells turbine with NACA0015 airfoil

The power P_b transferred from the flow to the blades composing the rotor is provided by Eq. (9), relying on Eqs. (5) and (8). The contribution of each blade element is integrated along the blade span, from hub to tip

$$P_b = \int_{r_{hub}}^{r_{tip}} Z \Omega r_i df_{\theta C_{IF,i}} = \frac{1}{2} \int_{r_{hub}}^{r_{tip}} Z \rho_{a,i} l_i C_{\theta C_{IF,i}} w_{1,i}^2 \Omega r_i dr \quad (9)$$

where Ω is the rotational speed of the rotor and Z is the number of row blades.

The static-to-static pressure drop across the rotor is provided by Eq. (10), which imposes the balance between the pressure force acting on the flow and the axial force acting on the blades

$$p_{1,i} - p_{2,i} = \frac{\rho_{a,i} l_i Z C_{x,i} v_{x1,i}^2}{4\pi r_i \sin^2 \beta_{1,i}} \quad (10)$$

where p is the static pressure, v_x is the axial component of the absolute velocity of the flow, and subscript 2 refers to the rotor outlet.

The flow absolute angle α_2 at the exit of the rotor is calculated with Eq. (11) to determine the velocity triangle at the domain outlet. Subsequently, all the components of the absolute and relative flow velocities and the other flow angles are computed by imposing the mass conservation and applying trigonometry

$$\cot \alpha_{2,i} = \cot \alpha_{1,i} + \frac{Z l_i C_{\theta,i}}{4\pi \beta_{1,i}} \quad (11)$$

The law of the absolute axial velocity distribution at the domain outlet is obtained with Eq. (12) by assigning the conditions of radial equilibrium and constant total pressure along the blade span [17]

$$\frac{d(v_{x,i}^2)}{dr} = -\frac{1}{4\pi^2 r_i^2} \frac{d(2\pi r_i v_{\theta 2,i})^2}{dr} \quad (12)$$

An iterative process based on energy conservation is implemented to calculate the airflow field in the entire computational domain [29,31].

4 Computational Fluid Dynamics Models

Three-dimensional CFD models were developed to analyze the aerodynamics of the airflow passing through a monoplane isolated Wells turbine for oscillating water column systems. The creation of the computational domain, constituted by the straight annular duct embedding the turbine, and its spatial discretization were performed with ANSYS ICEM 19.1 and implemented in ANSYS FLUENT 19.1 developing a completely automated procedure. Both steady and

unsteady Reynolds-averaged Navier–Stokes (RANS and u-RANS, respectively) schemes were used. The turbulence closure was performed with either the realizable $k-\epsilon$ model with enhanced wall functions or the $k-\omega$ stress transport model with a low Reynolds number correction to the turbulent viscosity. Two different approaches were utilized for modeling rotation: the multiple reference frame (MRF) model with a moving reference frame and the sliding mesh (SM) model [36].

4.1 Numerical Domain. The flow conditions and geometry of the straight annular duct and the turbine are axisymmetric in function to the number of rotor blades. Thus, the model is periodic in the circumferential direction. This enables to perform simulations on a computational domain comprising 1/7 of the turbine annulus, including one blade passage instead of analyzing the full annulus. As a consequence, the computational time and power required are significantly reduced. The domain is bounded by the inlet and outlet boundaries, the periodic surfaces, the annulus walls, and the blade walls (Fig. 3). The tip clearance gap was not included in the domain to limit the computational requirements. The implications of this geometric parameter on the turbine performance are determined by the extension of the gap. In particular, the decrease of its dimension enhances the turbine efficiency due to the reduction of the leakage losses but advances the aerodynamic stall. Therefore, the gap is typically defined between around 1% and 2% of the length of the blade chord to achieve a good compromise between these opposing effects. For such values, the gap influence on the operating curves of the turbomachine is limited [21–24].

The computational domain consists of three cell zones disposed in the axial direction: upstream duct, rotor, and downstream duct. The first and last regions are stationary and represent the annular duct sections upstream and downstream of the rotor. In contrast, the second is the rotating region enclosing one rotor blade in the annulus section. The domain is symmetrical in the axial direction as simulations imposing a sinusoidal function at the inlet were performed to define a bidirectional airflow [36]. The axial dimension of the stationary zones was set to six times the length of the blade chord to obtain the full development of the flow upstream of the rotor and examine the flow field after the interaction with the rotor blades [37,38]. For the rotating zone, the axial dimension was defined as equal to about the length of the blade chord according to the sensitivity analysis explained in Subsec. 4.5 for ensuring mesh independence.

4.2 Numerical Grid. The spatial discretization of the computational domain was achieved with a fully mapped mesh composed

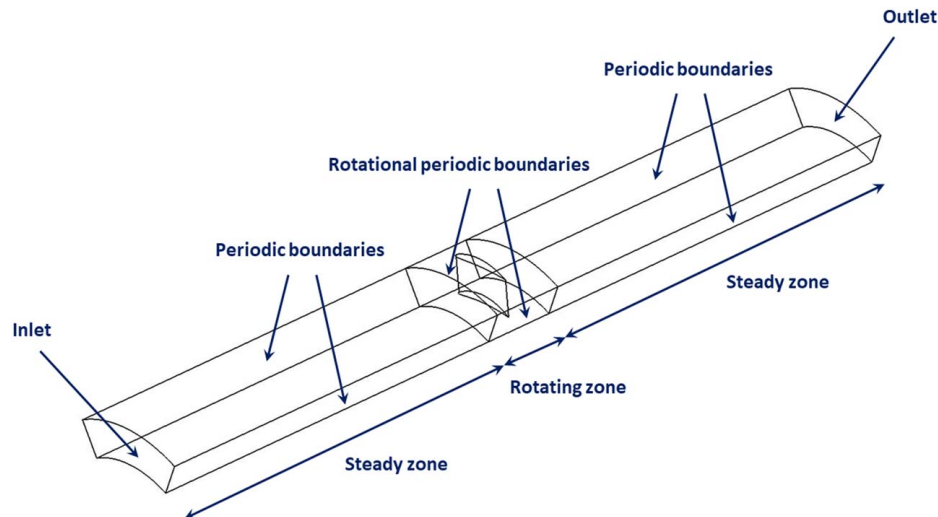


Fig. 3 Scheme of the computational domain of the monoplane isolated Wells turbine

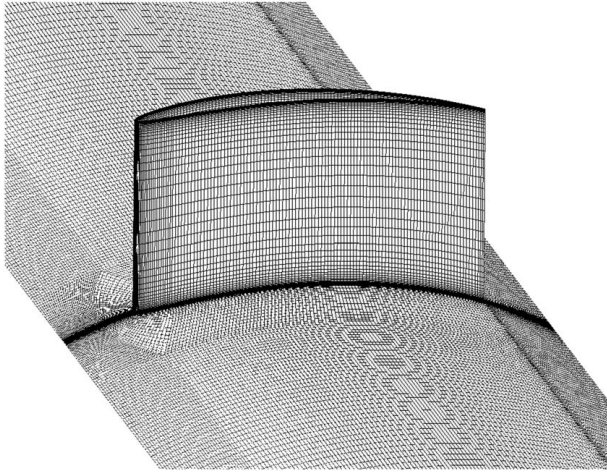


Fig. 4 Isometric view of the blade and hub wall mesh

of hexahedral elements by applying a multi-block technique and grid refinement at the wall boundaries [36]. A C-type mesh topology was realized in the region surrounding the blade to capture the complex boundary layer flow, and H-type mesh topologies were created in the rest of the domain (Fig. 4).

The mesh comprises around 3.85×10^6 nodes and 3.76×10^6 elements. The generated high-quality mesh led to well-converged solutions, characterized by the convergence criteria of the mass conservation within 10^{-3} and maximum order of the root mean squares residuals of 10^{-6} for the continuity, momentum, and turbulence equations. In the circumferential direction, 300 nodes were distributed around the blade profile, and 70 nodes were positioned in the blade passage with 30 nodes upstream of the leading edge and 40 downstream of the trailing edge. In the radial direction, 80 nodes were set along the blade span. In the axial direction, 80 nodes were located in the stationary regions, and 100 nodes were established in the rotating region. The dimension of the boundary layer cells in the direction normal to the wall boundary was defined to achieve values of y^+ between 30 and 90 for the simulations carried out with the $k-\varepsilon$ turbulence model and lower than 1 for the simulations performed with the $k-\omega$ turbulence model [36]. The cell growth ratio was set to 1.05 at all the wall boundaries (Fig. 5).

4.3 Mathematical Model. Steady and unsteady Reynolds-averaged Navier-Stokes equations were solved for a compressible ideal gas. The air density, the thermal conductivity, and the dynamic viscosity were defined as functions of the temperature [36]. The governing equations for the mass, the momentum, and the energy are described in equations from Eqs. (13)–(15)

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{v}) = 0 \quad (13)$$

where t is the time and $\mathbf{v}$ is the air velocity vector.

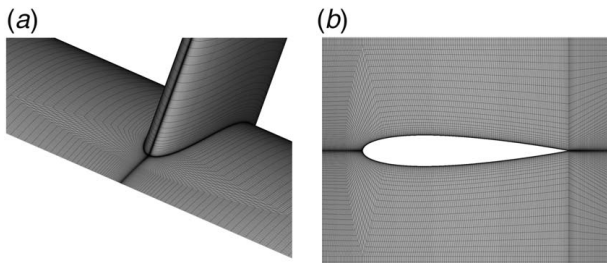


Fig. 5 Mesh close-up (a) at the hub leading edge and (b) around the blade tip

$$\frac{\partial(\rho \mathbf{v})}{\partial t} + \nabla \cdot (\rho \mathbf{v} \mathbf{v}) = -\nabla \cdot \mathbf{p} + \nabla \cdot \boldsymbol{\tau} \quad (14)$$

where $\boldsymbol{\tau}$ is the stress tensor.

$$\frac{\partial(\rho h_0)}{\partial t} + \nabla \cdot (\rho \mathbf{v} h_0) = \frac{\partial p}{\partial t} + \nabla \cdot (\kappa \nabla T + \boldsymbol{\tau} \cdot \mathbf{v}) \quad (15)$$

where h_0 is the total specific enthalpy, κ is the effective thermal conductivity, and T is the temperature.

The total specific enthalpy is provided by the specific enthalpy h and the specific kinetic energy, depending on the air velocity magnitude v , as expressed by Eq. (16)

$$h_0 = h + \frac{v^2}{2} \quad (16)$$

The time derivatives in the governing equations from Eqs. (13)–(15) are neglected in the steady-state analyses.

The Navier–Stokes equations were discretized with a second-order scheme for the pressure and second-order upwind schemes for the density, momentum, energy, turbulent kinetic energy, and specific dissipation rate. The coupled scheme was applied for relating the velocity and pressure corrections. The least-squares and cell-based method was utilized to calculate the variable gradients. Since the relative Reynolds number computed regarding the blade chord length ranges between around $2 \cdot 10^4$ and $3 \cdot 10^6$, the turbulence closure [21,39] was achieved with second-order discretization by using the realizable $k-\varepsilon$ model with enhanced wall functions for the near-wall treatment [40] or the $k-\omega$ stress transport model [41] with low Reynolds number correction to the turbulent viscosity [41]. The equations were implicitly solved with a pressure-based approach [36].

4.4 Boundary Conditions. The conditions imposed at the boundaries of the computational domain of Fig. 3 are listed in Table 2.

For all the numerical configurations, uniform distributions of the flow velocity and uniform and constant temperature distributions were assigned at the inlet boundary. At the domain outlet, uniform and constant distributions of the static pressure with radial equilibrium condition and a constant backflow temperature were imposed [36]. The rotational speed and the interval of the inlet velocity designated are provided in Table 1, while the static gauge pressure and the temperature were set to 0 Pa and 288.15 K, respectively.

The turbulence intensity ti was determined with Eq. (17) as a function of the relative Reynolds number $Re_{e,dh}$ computed on the basis of the hydraulic diameter of the annular duct [42]

$$ti = 0.16 Re_{e,dh}^{-0.125} \quad (17)$$

Table 2 Boundary conditions applied in the CFD models

Boundary	Condition
Inlet	Velocity, turbulent intensity, length scale, and temperature
Outlet	Static pressure, radial equilibrium, and turbulent intensity, length scale and temperature of the backflow
Blade and walls	No slip
Rotor circumferential sidewalls	Rotational periodicity
Duct circumferential sidewalls	Periodicity

The turbulent length scale l was defined through Eq. (18) based on the characteristic length of the flow opening L [42]

$$l = 0.07L \quad (18)$$

No-slip boundary conditions were imposed on the walls of the annulus and the blade. Moreover, periodic and rotationally periodic boundary conditions were assigned to the circumferential sidewalls of the stationary and rotating regions, respectively.

Regarding the uniform distributions of the inlet flow velocity, two different approaches were defined. Indeed, the effects of applying a discrete set of constant values covering the entire operating range and values varying with a sinusoidal function in the same range were analyzed. The first approach was used for both steady and unsteady simulations, while the second approach requires unsteady analyses.

The rotation was modeled with the multiple reference frame method or the sliding mesh method. In either case, a constant rotational speed was considered [36].

4.5 Sensitivity Analyses. The development of a computational fluid dynamics model requires sensitivity analyses with respect to spatial and temporal discretization.

For evaluating the grid independence, a mesh sensitivity analysis was performed by comparing the results of several different grids with a node number ranging from 0.5 to 9.5×10^6 . The finer and coarser meshes were derived through homogeneous variations of the node number relative to the selected mesh comprising 3.85 million nodes. The minimum values of node number around the blade airfoil and along its span were set to 200 and 50 , respectively.

The mesh sensitivity analysis consists of three phases. First, the effects of the number of elements and their distribution on the calculated values were assessed over the entire computational domain. Afterward, the influence of the extension in the axial direction of the rotating cell zone was evaluated to determine the minimum value ensuring mesh independence. The ultimate phase regarded the extensive appraisal of the implications of the element number and distribution on the computed quantities [36].

The number and arrangement of elements are fundamental features, in particular for the rotating cell zone. As noticeable in Fig. 6(a), an extension of at least about half a chord both upstream and downstream of the leading edge of the blade ensures the independence of the computations and thus of the rotor torque from this parameter. Figure 6(b) indicates that the difference of the torque calculated corresponding to the maximum inlet velocity is 5.8% for a grid with approximately 0.5×10^6 nodes compared to the finest mesh with about 9.5×10^6 nodes. The discrepancy reduces to 1.0% for a mesh with approximately 1.6×10^6 nodes. By further

increasing the node number, the variations become slight, and for a grid with around 3.85 million nodes are negligible.

The steady-state simulations with the $k-\omega$ turbulence model and the multiple reference frame method were selected to present the outcomes of the mesh sensitivity analysis. The selected quantity, flow conditions, and numerical settings are representative of the effects of the investigated grid parameters for all the CFD analyses performed.

The definition of the values of the time-step for fully transient models requires a sensitivity analysis. The objective is to evaluate the effects of the magnitude of the time-step for determining the highest value that ensures slight or negligible variations of the computed quantities. Values of the time-step size lower or equal to $1 \cdot 10^{-2}$ s provide a satisfactory prediction of the torque during the entire computations for the multiple reference frame model. For the sliding mesh model, such value entails fluctuations of the computed torque corresponding to the extremes of the inflow and outflow phases, which are avoided by a time-step less or equal to $8 \cdot 10^{-3}$ s. A further decrease of the time-step size to at least $5 \cdot 10^{-3}$ s is needed for reaching discrepancies of the computed results lower than 0.25% and 0.40% for the MRF and the SM models, respectively (Fig. 7). Therefore, this value was utilized in further computations.

For the stationary analyses, the duration of the calculations was normalized through the ratio with that of the simulations with the MRF scheme and 3.85×10^6 nodes to compare the computational time of each case. For the transient analyses, the case with the same rotational scheme and mesh and the time-step size of $5 \cdot 10^{-3}$ s was considered as a reference for normalization. Figure 8(a) shows the trade-off between the accuracy of the calculations of the static-to-static pressure drop across the turbine duct and the computational time to changes in the grid size for the steady-state calculations with the $k-\omega$ turbulence model and the MRF method. Figure 8(b) indicates that the computational time of the SM approach is always higher regarding the MRF approach. In particular, for the selected time-step size of $5 \cdot 10^{-3}$ s, the computational time is about 19% greater [36].

Since the study aimed to investigate different CFD models, a fine grid with around 3.85×10^6 nodes and the time-step size of $5 \cdot 10^{-3}$ s were selected to achieve high accuracy while limiting the computational time.

5 Results and Discussion

The results of the aerodynamics of the Wells turbine achieved with the BEM model and the CFD models are presented in terms of nondimensional and dimensional parameters, as traditionally

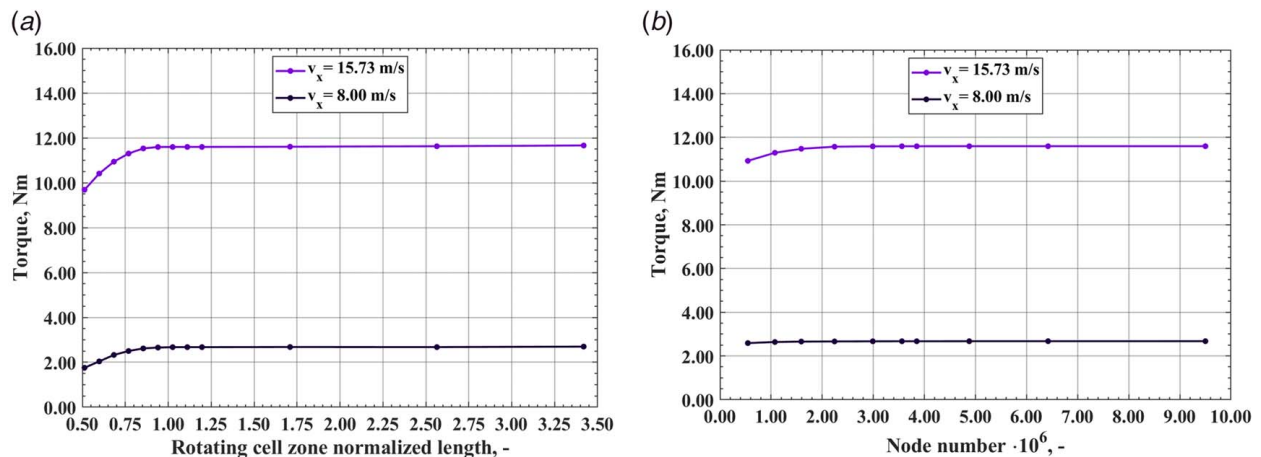


Fig. 6 Torque acting on the rotor blades for different values of the inlet flow velocity as a function of (a) the length of the rotating cell zone normalized relative to the blade chord and (b) the grid node number

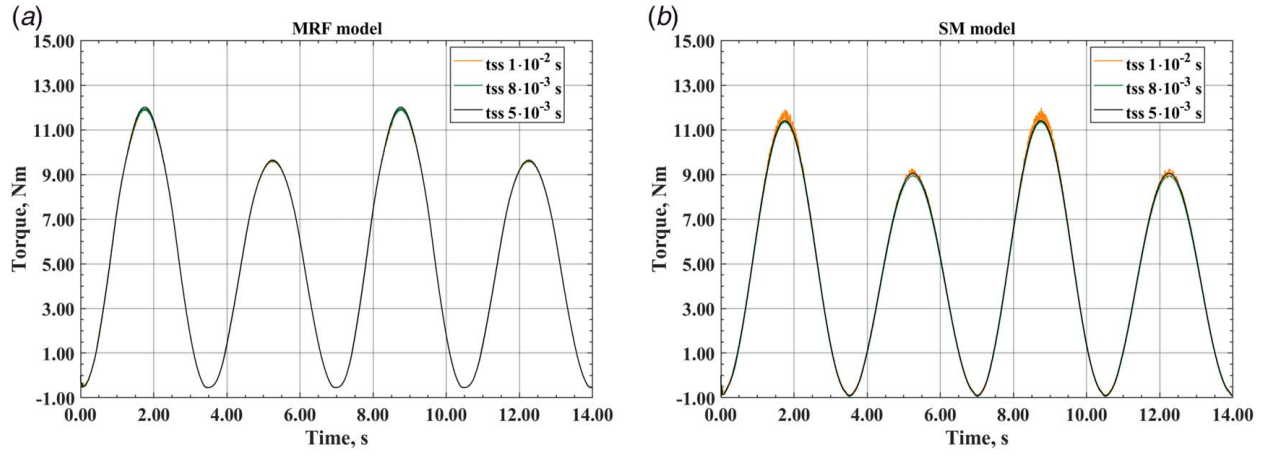


Fig. 7 Sensitivity analysis of the blade torque to the time-step size for (a) the MRF model and (b) the SM model

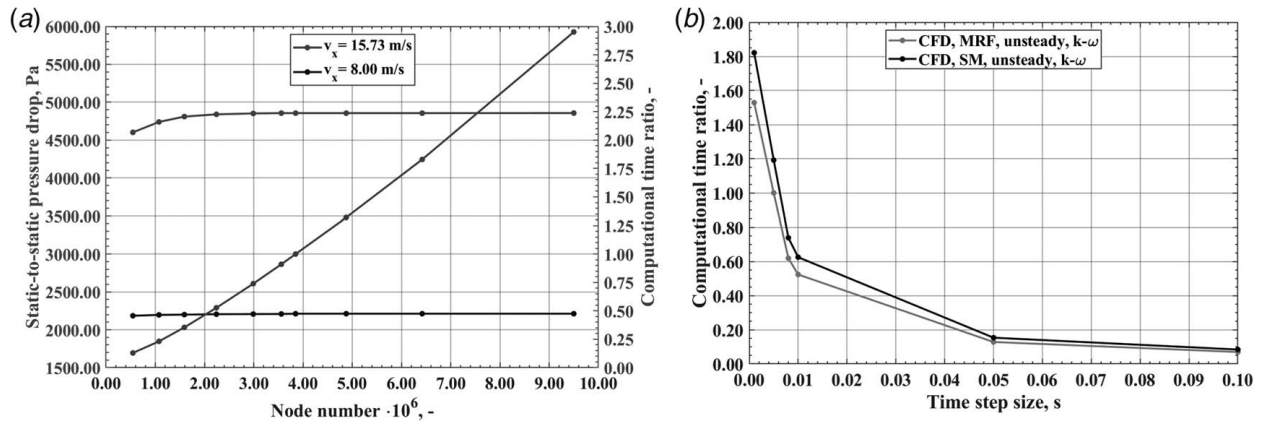


Fig. 8 Computational time ratio for (a) the steady-state analyses with the MRF model as a function of the node number and (b) the transient MRF and SM models as a function of the time-step size

applied in turbomachinery [34]. The nondimensional parameters are the flow coefficient φ , the input and torque coefficients C_A and C_T , and the efficiency η , defined in Eqs. (19)–(22), respectively. The dimensional parameters are the torque, the power, and the static-to-static pressure drop across the computational domain

$$\varphi_m = \frac{v_{x,m}}{U_m} \quad (19)$$

$$C_{A,m} = \frac{(p_{01} - p_2)\dot{q}}{\frac{1}{2}\rho_1(v_{x,m}^2 + U_m^2)blZv_{x,m}} \quad (20)$$

$$C_{T,m} = \frac{T_b}{\frac{1}{2}\rho_1(v_{x,m}^2 + U_m^2)blZr_m} \quad (21)$$

$$\eta = \frac{\Omega T_b}{(p_{01} - p_2)\dot{q}} = \frac{C_{T,m}}{\varphi_m C_{A,m}} \quad (22)$$

where U is the blade speed, p_{01} is the total pressure at the domain inlet, p_2 is the static pressure at the domain outlet, $\dot{q}$ is the air volume flowrate, b is the blade height, T_b is the rotor blade torque, ρ_1 is the air density at the domain inlet, and subscript m refers to the blade midspan.

Figure 9(a) shows that the input coefficient has a linear relation with the flow coefficient and deviates from this trend only at the extremes of the operating range. The deviation is very slight for the CFD simulations with the $k-\omega$ turbulence model, while it is more marked for the cases with the $k-\varepsilon$ model due to the prediction of greater pressure drops across the turbine duct. For the analytical model, the linear relation discrepancy is higher owing to the lower values of the pressure variation.

In Fig. 9(b), the torque coefficient is negative for low incidence angles, corresponding to the values of φ_m between approximately -0.05 and 0.05 . This occurs as the contribution of the viscous forces prevails on that of the pressure forces acting on the surfaces of the rotor blades. For higher absolute values of the flow incidence angle, the torque is positive, as the contribution of the pressure forces is dominant. The BEM model provides higher levels of the torque coefficient regarding the CFD models due to the greater blade torque predicted.

Figure 10(a) shows the static-to-static pressure drop between the turbine inlet and outlet as a function of the volume airflow rate. For all the models, the relation is linear and deviates from linearity only at the extremes of the flow phases.

The torque acting on the rotor blades is represented in Fig. 10(b) by varying the midspan flow coefficient. The torque increases with the rise of the flow incidence angle.

In Fig. 10(c), the power transferred by the flow to the blades is shown. Since the rotational velocity is constant, the power curves

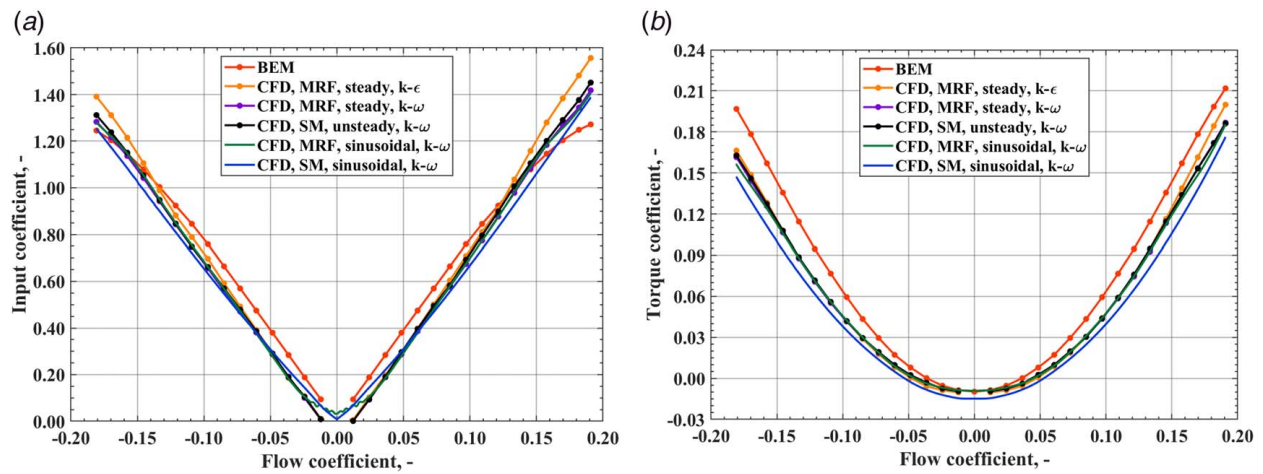


Fig. 9 (a) Input coefficient and (b) torque coefficient versus flow coefficient at the blade midspan

have the same trend as those of the torque. It is noticeable that the results of the CFD models are in close agreement with the experimental literature results for the whole inflow phase and the majority of the outflow phase. Conversely, at the extreme of the outflow phase, the experimental curve of the power is subject to a reduction of the growth speed, which is not shown by the CFD models [26,32]. In particular, the $k-\omega$ turbulence model predicts power values closer to the experimental results than the $k-\epsilon$ model. The overestimation of the power of the numerical models can be

addressed to the absence of mechanical losses and, for the CFD models, even of the tip clearance. The BEM model provides results in excellent agreement with the analytical literature model and has a broader range of applicability than this reference, as it considers both the inflow and outflow phases. Either the analytical models overestimate the rotor power.

The curves of efficiency are depicted in Fig. 10(d). The turbine efficiency increases with the absolute value of the flow coefficient up to a specific value, after which it declines. The peak efficiency

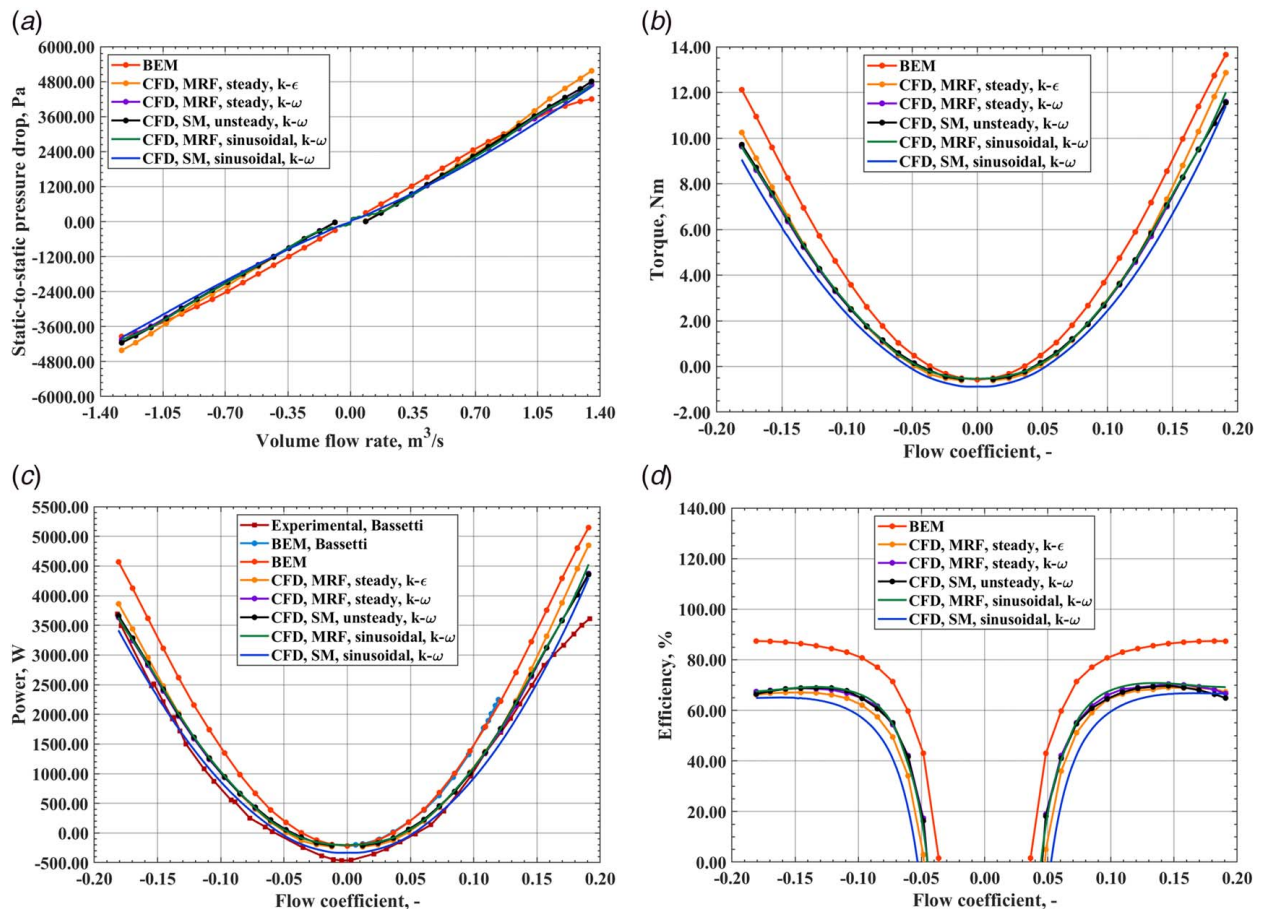


Fig. 10 (a) Static-to-static pressure drop versus volume flowrate and (b) torque, (c) power, and (d) efficiency as functions of the flow coefficient at the blade midspan

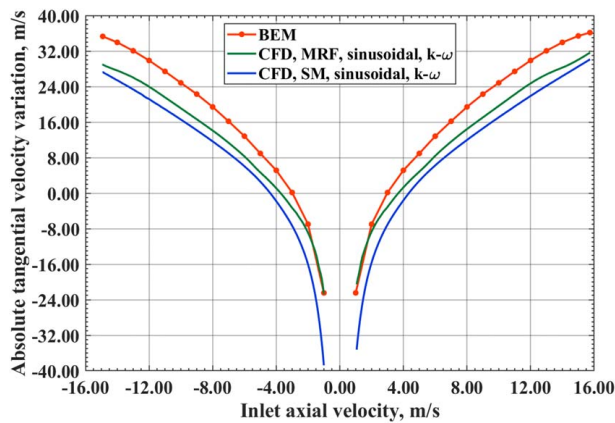


Fig. 11 Absolute tangential velocity versus inlet axial velocity

is reached during the outflow phase and is nearly 87% for the BEM model and ranges between 69% and 71% for the CFD models. The discrepancy in the efficiency calculated by the analytical model is mainly due to the overestimation of the aerodynamic interference between the blades comprising the rotor.

The CFD models based on applying a sinusoidal function for the inlet velocity and the $k-\omega$ turbulence model were selected to examine further the modeling, as they provide the closest results to the experimental data.

In Fig. 11, the variation of the absolute tangential velocity with the airflow is analyzed. Since the flow entering the turbine duct is assumed to be perfectly axial, this quantity coincides with the tangential component of the absolute velocity in the downwind section. In particular, for the analytical model, the variation is calculated between the inlet and outlet sections positioned far upstream and far downstream of the rotor. For the CFD models, the maximum

average value of the change in absolute tangential velocity inside the computational domain is considered. This value is reached right downwind of the rotor blades, as moving toward the outlet the swirl decreases.

The variations of the absolute tangential velocity achieved with the BEM model are generally higher regarding the CFD models in the entire operating range, except at the lowest absolute values of the flow coefficient for the case of the MRF model. The differences in the outcomes are due to the overprediction by the analytical model of the tangential force acting on the rotor blades. This, in turn, depends on the overestimation of the lift force regarding the drag force.

Focusing on the CFD models, the flow fields of the static-to-static pressure drop across the turbine are shown in Fig. 12. Since the analyses were performed on a section of the turbine annulus by imposing circumferential periodicity boundary conditions instead of solving the computations for the full annulus, the influence of the potential effect on the rotor blades was investigated. This effect is caused by the interaction of the wake at the trailing edge of each blade with the leading edge of the following blade of the row. Figure 12 indicates that the potential effect is limited during both the flow phases. Thus, the application of circumferential periodicity is a reasonable approach. The flow field depicted at the midspan is representative of the entire span, and the predictions of the two schemes for modeling rotation are in close agreement.

Figures 13 and 14 show the flow fields of the absolute tangential velocity computed with the MRF model and the SM model on evenly spaced surfaces with a constant radius at 25%, 50%, and 75% of the blade span for the inflow and outflow phases. The flow is attached to the blade from the hub to the tip for all the investigated conditions. Higher velocities can be noticed moving toward the blade tip, in particular close to the leading edge, due to the higher peripheral speed.

The flow fields obtained with the MRF and SM models are in good agreement. However, the average variation of the absolute

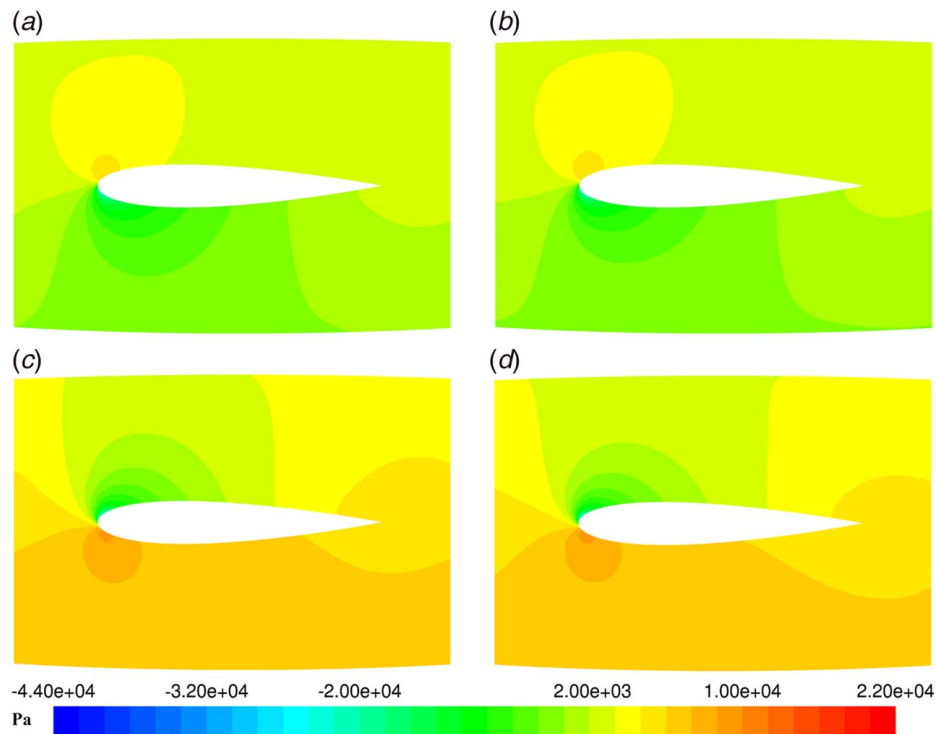


Fig. 12 Relative static pressure on surfaces with a constant radius at the blade midspan for (a) the inflow phase with the MRF model, (b) the inflow phase with the SM model, (c) the outflow phase with the MRF model, and (d) the outflow phase with the SM model

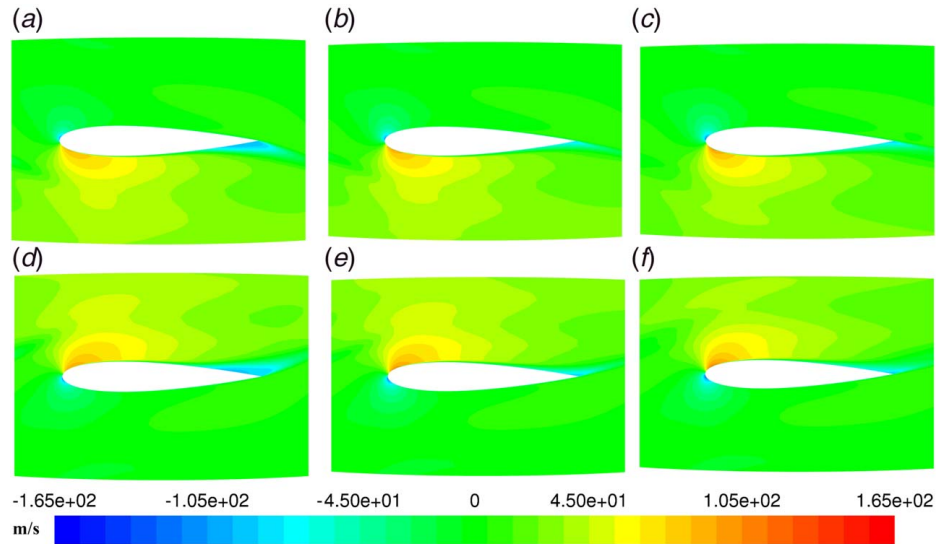


Fig. 13 Absolute tangential velocity computed with the MRF model on surfaces with a constant radius at 25%, 50%, and 75% of the span for (a)–(c) the inflow phase and (d)–(f) the outflow phase

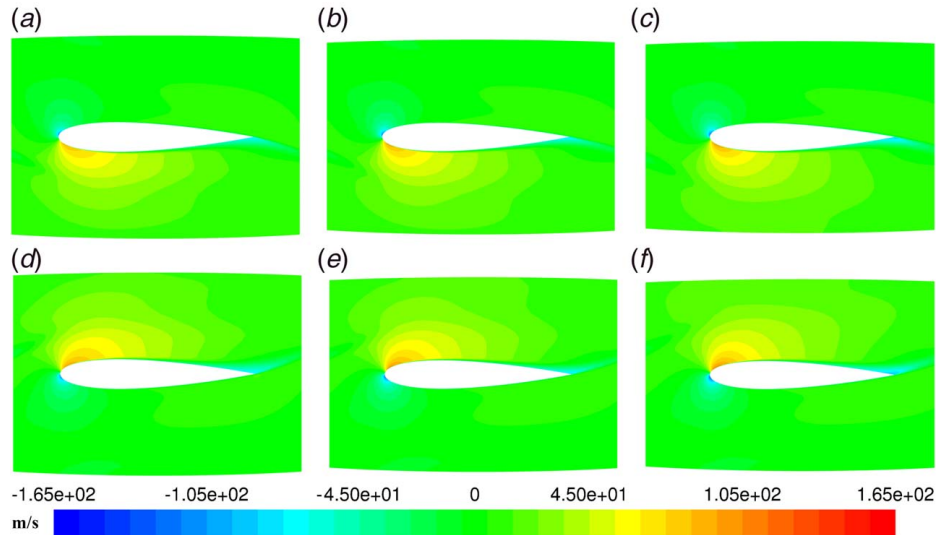


Fig. 14 Absolute tangential velocity computed with the SM model on surfaces with a constant radius at 25%, 50%, and 75% of the span for (a)–(c) the inflow phase and (d)–(f) the outflow phase

tangential velocity is greater for the former model due to higher absolute values of the minimum and maximum absolute tangential velocity. This results in the prediction of higher values of torque and power, as shown in Figs. 10(b) and 10(c), respectively.

The reduction of the absolute tangential velocity in the wake at the outlet of the blade is presented in Fig. 15 on sections evenly spaced in the circumferential direction in correspondence with the midspan radial position. Moving away from the blade, from line 1 to line 5, the effects of the wake are reduced, the wake tends to mix with the surrounding flow, and it is deflected in the downstream direction. The influence of the wake is more pronounced during the outflow than the inflow, owing to the different inlet air conditions. The predicted trend of the absolute tangential velocity is similar for the MRF and SM models, although the variations obtained with the former are higher.

In Fig. 16, the distributions of the isentropic Mach number and the static pressure are represented along the curvilinear abscissa on the airfoil at the blade midspan. The isentropic Mach

number is obtained with Eq. (23) by applying the isentropic flow relations

$$Ma_{is} = \sqrt{\left(\left(\frac{p_{0,fr}}{p} \right)^{\frac{\gamma-1}{\gamma}} - 1 \right) \frac{2}{\gamma-1}} \quad (23)$$

where γ is the heat capacity ratio of air and $p_{0,fr}$ is the reference total pressure calculated based on the inlet static pressure and the inlet velocity computed in the rotating reference frame.

At the leading edge of the suction side, the flow has a rapid acceleration and quickly reaches the maximum isentropic Mach number corresponding to the minimum of the static pressure. Afterward, the flow experiences a smooth deceleration to its lowest value due to the airfoil shape. On the pressure side, the flow first decelerates and then accelerates with a gradual trend in the region subject to the slow reduction of the static pressure. During the inflow, the static

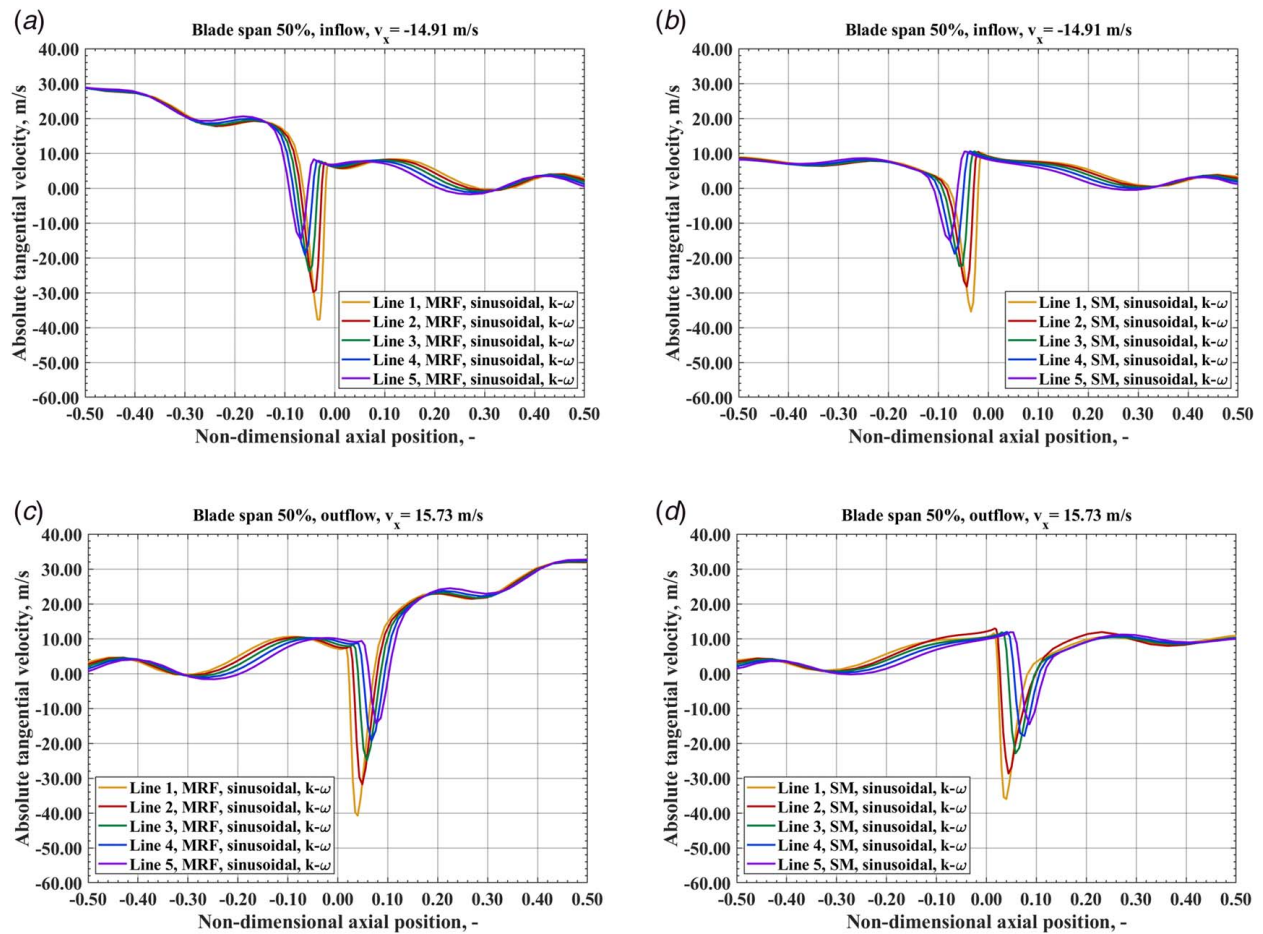


Fig. 15 Distributions of the absolute tangential velocity in the trailing edge region computed during (a) and (b) the inflow phase and (c) and (d) the outflow phase with the MRF and SM models

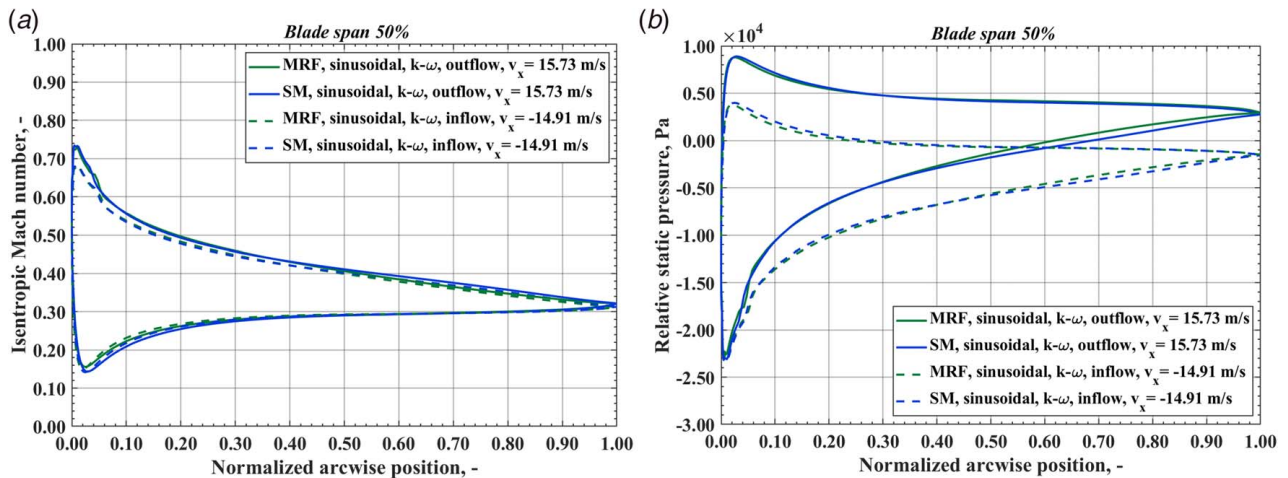


Fig. 16 Distributions of (a) the isentropic Mach number and (b) the relative static pressure along the blade airfoil at the midspan section

pressure on the blade surfaces is lower regarding the outflow and results in smaller variations of the isentropic Mach number.

Figure 17 indicates the variation of the specific enthalpy between the inlet and outlet of the turbine duct as a function of the specific entropy. The variation in the specific total enthalpy equals the difference between the specific work exchanged by the flow with the rotor blades and the specific work lost for the friction with the other wall boundaries. The specific total enthalpy drop during the outflow is greater than during the inflow, resulting

in a higher power transferred from the airflow to the rotor. Considering the entire operating range, the variation in the specific total enthalpy determines an average power on the rotor blades that is 6.09% lower for the SM model regarding the MRF model.

The values of the power transferred from the flow to the turbine and the efficiency computed with the analytical and numerical models are compared in Table 3 considering the extremes of the flow phases.

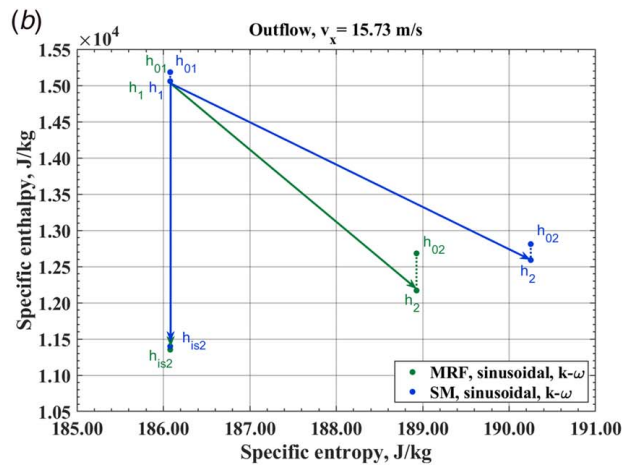
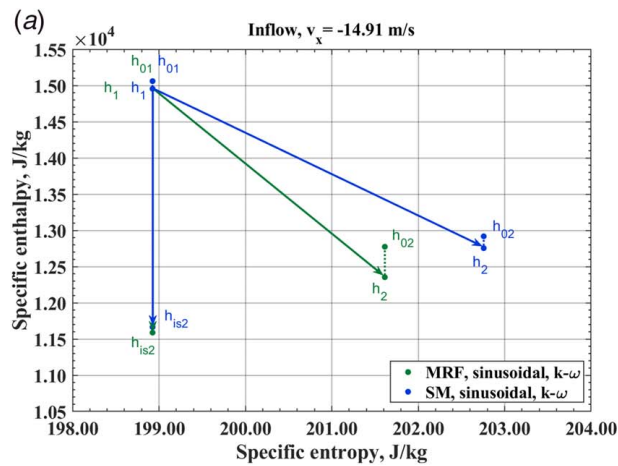


Fig. 17 Specific enthalpy variation versus specific entropy generation across the turbine duct calculated with (a) the MRF model and (b) the SM model

Table 3 Power conversion and efficiency computed with the BEM and CFD models

Quantity	Inflow			Outflow		
	BEM	MRF	SM	BEM	MRF	SM
v_{x1} , m/s	-14.91	-14.91	-14.91	15.73	15.73	15.73
P_t , W	5243.64	5367.45	5252.39	5915.37	6541.78	6453.66
P_b , W	4568.33	3625.09	3411.34	5146.60	4527.93	4300.17
η_b , %	87.12	67.54	64.95	87.00	69.22	66.63

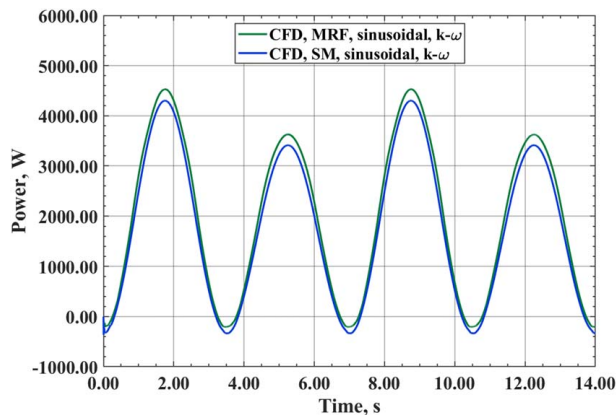


Fig. 18 Time series of the power exerted by the airflow on the rotor blades determined with the MRF and SM models

Finally, the time series of the power converted by the monoplane isolate Wells turbine are represented in Fig. 18 for the CFD models with a sinusoidal function for the inlet velocity.

6 Conclusions

The operation of a Wells turbine for oscillating water column systems was analyzed with an analytical model and computational fluid dynamics models. The former modeling typology applies a theoretical approach to solve the aerodynamics, considering the cascades as groups of blades composed of a finite set of airfoils. The model combines satisfactory accuracy and limited computational effort. Therefore, it is especially suitable during the early design stages to provide a rapid assessment of the energy extractable by a Wells turbine. The latter modeling typology provides higher

accuracy but requires significantly higher computational resources. Thus, numerical models are useful for investigating in detail a limited number of selected configurations. The domain of the duct comprising the Wells rotor was generated with a three-dimensional multi-block technique, obtaining a fully mapped mesh composed of hexahedral elements. Circumferential periodic boundary conditions were applied to reduce the computational power and time requested. The CFD models use the RANS or u-RANS schemes with a moving reference frame or the u-RANS formulation with a sliding mesh.

The blade element momentum model provides results that are in excellent agreement with the analytical literature model. Moreover, the developed model has a wider range of applicability as it considers either the inflow phase and the outflow phase. By comparing this model with the experiments, reasonable matching of the power output curve is obtained with a limited overestimation of the outcomes, also due to the absence of mechanical losses. As regards the numerical models, the achieved results are in good agreement with the experimental literature data.

The computational model providing the closest results to the experimental data applies the sliding mesh scheme and the $k-\omega$ turbulence model and assigns a sinusoidal function for the inlet velocity. This configuration and the use of a mesh with around 3.85×10^6 nodes and a time-step of $5 \cdot 10^{-3}$ s are suggested to achieve high accuracy. Moreover, time-variant inlet boundary conditions allow the analysis of the flow quantities and the performance parameters in the time domain, making the CFD model suitable for coupling with the models of the primary and tertiary converters. Conversely, when the objective of the analysis is comparing a wide variety of cases, the frozen rotor approximation (MRF model) is a cost-effective approach to evaluating the machine operation and a mesh with about 1.6×10^6 nodes and a time-step size up to $1 \cdot 10^{-2}$ s may be utilized. For further speeding up the computations, the use of incompressible gas formulations is recommended.

The analytical and numerical models were applied to determine the operating curves of a laboratory-scale turbine. Subsequently, the flow fields and the distributions of the main quantities were analyzed in detail. Finally, the CFD models were utilized to provide the time series of the power output of the Wells turbine in environmental conditions typical of the Mediterranean Sea.

Acknowledgment

Part of the research leading to the results presented received funding from the European Union and the Republic of Poland in the framework of the NAWA Project with the Grant Agreement 10/P/2019.

Conflict of Interest

There are no conflicts of interest.

Data Availability Statement

The data sets generated and supporting the findings of this article are obtainable from the corresponding author upon reasonable request. The authors attest that all data for this study are included in the paper. Data provided by a third party were listed in Acknowledgment. No data, models, or code were generated or used for this paper.

References

- [1] Mork, G., Barstow, S., Kabuth, A., and Pontes, T. M., 2010, "Assessing the Global Wave Energy Potential," Proceedings of the 29th International Conference on Ocean, Offshore Mechanics and Arctic Engineering, Shanghai, China, June 6–11.
- [2] Gunn, K., and Stock-Williams, C., 2012, "Quantifying the Global Wave Power Resource," *Renewable Energy*, **44**, pp. 296–304.
- [3] BP, 2019, "Statistical Review of World Energy," 68th ed., <https://www.bp.com/content/dam/bp/business-sites/en/global/corporate/pdfs/energy-economics/statistical-review/bp-stats-review-2019-full-report.pdf>, Accessed January 20, 2021.
- [4] Curran, R., Whittaker, T. J. T., and Stewart, T. P., 1998, "Aerodynamic Conversion of Ocean Power From Wave to Wire," *Energy Convers. Manage.*, **39**(16–18), pp. 1919–1929.
- [5] Falcão, A. F. O., 2010, "Wave Energy Utilization: A Review of the Technologies," *Renewable Sustainable Energy Rev.*, **14**(3), pp. 899–918.
- [6] Melikoglu, M., 2018, "Current Status and Future of Ocean Energy Sources: A Global Review," *Ocean Eng.*, **148**, pp. 563–573.
- [7] Falcão, A. F. O., Henriques, J., and Gato, L., 2018, "Self-rectifying Air Turbines for Wave Energy Conversion: A Comparative Analysis," *Renewable Sustainable Energy Rev.*, **91**, pp. 1231–1241.
- [8] Wells, A. A., 1976, "Fluid Driven Rotary Transducer," British Patent No. 1595700.
- [9] Gato, L. M. C., and Falcão, A. F. O., 1984, "On the Theory of the Wells Turbine," *ASME J. Eng. Gas Turbines Power*, **106**(3), pp. 628–633.
- [10] Inoue, M., Kaneko, K., Setoguchi, T., and Shimamoto, K., 1986, "Studies on Wells Turbine for Wave Power Generator," *J. Soc. Mech. Eng.*, **29**(250), pp. 1177–1182.
- [11] Maeda, H., Santhakumar, S., Setoguchi, T., Takao, M., Kinoue, Y., and Kaneko, K., 1999, "Performance of an Impulse Turbine With Fixed Guide Vanes for Wave Power Conversion," *Renewable Energy*, **17**(4), pp. 533–547.
- [12] Setoguchi, T., Santhakumar, S., Maeda, H., Takao, M., and Kaneko, K., 2001, "A Review of Impulse Turbines for Wave Energy Conversion," *Renewable Energy*, **23**(2), pp. 261–292.
- [13] Setoguchi, T., Takao, M., Santhakumar, S., and Kaneko, K., 2004, "Study of an Impulse Turbine for Wave Power Conversion: Effects of Reynolds Number and Hub-to-Tip Ratio on Performance," *ASME J. Offshore Mech. Arct. Eng.*, **126**(2), pp. 137–140.
- [14] Raghunathan, S., Tan, C. P., and Ombaka, O. O., 1985, "Performance of the Wells Self-rectifying Air Turbine," *Aeronaut. J.*, **89**, pp. 369–379.
- [15] Raghunathan, S., Setoguchi, T., and Kaneko, K., 1991, "Aerodynamics of Monoplane Wells Turbine—A Review," Proceedings of the First International Offshore and Polar Engineering Conference, Edinburgh, Scotland, Aug. 11–16, Vol. 1, pp. 370–379.
- [16] Raghunathan, S., 1995, "A Methodology for Wells Turbine Design for Wave Energy Conversion," *Proc. Inst. Mech. Eng. Part A: J. Power Energy*, **209**(3), pp. 221–232.
- [17] Raghunathan, S., 1995, "The Wells Air Turbine for Wave Energy Conversion," *Prog. Aerosp. Sci.*, **31**(4), pp. 335–386.
- [18] Brito-Melo, A., Gato, L. M. C., and Sarmiento, A. J. N. A., 2002, "Analysis of Wells Turbine Design Parameters by Numerical Simulation of the OWC Performance," *Ocean Eng.*, **29**(12), pp. 1463–1477.
- [19] Heikal, H. A., Hafiz, A. A., Bayomi, N. N., and Ahmed, M. H., 2003, "Theoretical and Experimental Investigation on the Wells Turbine Performance," Proceedings of the 7th Conference on Theoretical and Applied Mechanics, Cairo, Egypt, Mar. 11–13.
- [20] Torresi, M., Camporeale, S. M., Pascasio, G., and Fortunato, B., 2004, "Fluid Dynamic Analysis of the Solidity of a Wells Turbine," Proceedings of the 59th Annual Conference ATI, Genova, Italy, Sept. 14–17.
- [21] Torresi, M., Camporeale, S. M., Strippoli, P. D., and Pascasio, G., 2008, "Accurate Numerical Simulation of a High Solidity Wells Turbine," *Renewable Energy*, **33**(4), pp. 735–747.
- [22] Taha, Z., Sugiyono, and Sawada, T., 2010, "A Comparison of Computational and Experimental Results of Wells Turbine Performance for Wave Energy Conversion," *Appl. Ocean Res.*, **32**(1), pp. 83–90.
- [23] Taha, Z., Sugiyono, Tuan Ya, T. M. Y. S., and Sawada, T., 2011, "Numerical Investigation on the Performance of Wells Turbine With Non-Uniform Tip Clearance for Wave Energy Conversion," *Appl. Ocean Res.*, **33**(4), pp. 321–331.
- [24] Torresi, M., Pranzo, D., Camporeale, S. M., and Pascasio, G., 2011, "Improved Design of High Solidity Wells Turbine," Proceedings of the 9th European Wave and Tidal Energy Conference, Southampton, UK, Sept. 5–9.
- [25] Starzmann, R., and Carolus, T. H., 2011, "Effect of Design Parameters on Aeroacoustic and Aerodynamic Performance of Wells Turbines," Proceedings of the 30th International Conference on Ocean, Offshore and Arctic Engineering, Rotterdam, The Netherlands, ASME.
- [26] Bassetti, M., Corsini, A., Delibra, G., and Rispoli, F., 2014, "Progetto e Prima Verifica Operativa di una μ -Turbina Wells per il Mar Mediterraneo," ENEA, (in English).
- [27] Suchithra, R., Ezhilsabareesh, K., and Samad, A., 2019, "Development of a Reduced Order Wave to Wire Model of an OWC Wave Energy Converter for Control System Analysis," *Ocean Eng.*, **172**, pp. 614–628.
- [28] Henriques, J. C. C., Portillo, J. C. C., Sheng, W., Gato, L. M. C., and Falcão, A. F. O., 2019, "Dynamics and Control of Air Turbines in Oscillating-Water-Column Wave Energy Converters: Analyses and Case Study," *Renewable Sustainable Energy Rev.*, **112**, pp. 571–589.
- [29] Ciappi, L., Cheli, L., Simonetti, I., Bianchini, A., Talluri, L., Cappietti, L., and Manfreda, G., 2020, "Analytical Models of Oscillating Water Column Systems Operating With Air Turbines in the Mediterranean Sea," Proceedings of the 15th Conference on Sustainable Development of Energy, Water and Environment Systems, Cologne, Germany, Sept. 1–5.
- [30] Ciappi, L., Cheli, L., Simonetti, I., Bianchini, A., Manfreda, G., and Cappietti, L., 2020, "Wave-to-Wire Model of an Oscillating-Water-Column Wave Energy Converter and Its Application to Mediterranean Energy Hot-Spots," *Energies*, **13**(21), p. 5582.
- [31] Ciappi, L., Cheli, L., Simonetti, I., Bianchini, A., Talluri, L., Cappietti, L., and Manfreda, G., 2021, "Wave-to-Wire Models of Wells and Impulse Turbines for Oscillating Water Column Wave Energy Converters Operating in the Mediterranean Sea," *Energy*, **238**, Part A, p. 121585.
- [32] Bassetti, M., Corsini, A., Delibra, G., Faggiolati, G., Piccinini, S., Rispoli, F., Romani, G., Ruggeri, M., and Tuccimei, E., 2012, "An Integrated Procedure for the Design of a Wells Turbine Developed for Mediterranean Operation," Proceedings of the 7th European Seminar on Offshore Wind and Other Marine Renewable Energies in Mediterranean and European Seas, Rome, Italy, Sept. 5–7.
- [33] Horlock, J. H., 1973, *Axial Flow Compressors, Fluid Mechanics and Thermodynamics*, RE Krieger Publishing Company, Malabar, FL.
- [34] Dixon, S. L., 2005, *Fluid Mechanics and Thermodynamics of Turbomachinery*, 5th ed., Elsevier, The Netherlands.
- [35] Raghunathan, S., Tan, C. P., and Ombaka, O. O., 1986, "Performance of Wells Self Rectifying Air Turbine," *J. R. Aeronaut. Soc.*, **89**, pp. 369–379.
- [36] Ciappi, L., Stebel, M., Smolka, J., Cappietti, L., and Manfreda, G., 2020, "Computational Fluid Dynamics Models of Wells Turbines for Oscillating Water Column Systems," Proceedings of the 6th International Conference on Contemporary Problems of Thermal Engineering, Cracow, Poland, Sept. 21–25.
- [37] Setoguchi, T., Kinoue, Y., Kim, T. H., Kaneko, K., and Inoue, M., 2003, "Hysteretic Characteristics of Wells Turbine for Wave Power Conversion," *Renewable Energy*, **28**(13), pp. 2113–2127.
- [38] Halder, P., Abdus, S., and Thevenin, D., 2017, "Improved Design of a Wells Turbine for Higher Operating Range," *Renewable Energy*, **106**, pp. 122–134.
- [39] Hashem, I., Abdel Hameed, H. S., and Mohamed, M. H., 2018, "An Axial Turbine in an Innovative Oscillating Water Column (OWC) Device for Sea-Wave Energy Conversion," *Ocean Eng.*, **164**, pp. 536–562.
- [40] Launder, B. E., and Spalding, D. B., 1972, *Lectures in Mathematical Models of Turbulence*, Academic Press, London, England.
- [41] Menter, F. R., 1994, "Two-Equation Eddy-Viscosity Turbulence Models for Engineering Applications," *Am. Inst. Aeronaut. Astronaut. J.*, **32**(8), pp. 1598–1605.
- [42] ANSYS Incorporated, 2019, "ANSYS Fluent Theory Guide," Southpointe 2600 ANSYS Drive, Canonsburg, PA.