

Effectiveness of two-dimensional CFD simulations for Darrieus VAWTs: a combined numerical and experimental assessment

Alessandro Bianchini¹, Francesco Balduzzi¹, Peter Bachant², Giovanni Ferrara¹, Lorenzo Ferrari^{3*}

¹ Department of Industrial Engineering, University of Florence - Via di Santa Marta 3, 50139, Firenze, Italy - Tel. +39 055 275 8773 - Fax +39 055 275 8755 - bianchini@vega.de.unifi.it

² Center for Ocean Renewable Energy, University of New Hampshire - Chase Ocean Engineering Laboratory, Durham, N.H. 03824 - pxl3@wildcats.unh.edu

³ Department of Energy, Systems, Territory and Construction Engineering, University of Pisa - Largo Lucio Lazzarino 1, 56122, Pisa, Italy - Tel. +39 050 221 7132 - Fax +39 050 221 7333 - lorenzo.ferrari@unipi.it

* = contact author

Abstract

Thanks to the continuous improvement of calculation resources, computational fluid dynamics (CFD) is expected to provide in the next few years a cost-effective and accurate tool to improve the understanding of the unsteady aerodynamics of Darrieus wind turbines. This rotor type is in fact increasingly welcome by the wind energy community, especially in case of small size applications and/or non-conventional installation sites.

In the present study, unique tow tank experimental data on the performance curve and the near-wake structure of a Darrieus rotor were used as a benchmark to validate the effectiveness of different CFD approaches. In particular, a dedicated analysis is provided to assess the suitability, the effectiveness and the future prospects of simplified two-dimensional (2D) simulations. The correct definition of the computational domain, the selection of the turbulence models and the correction of simulated data for the parasitic torque components are discussed in this study. Results clearly show that, (only) if properly set, two-dimensional CFD simulations are able to provide - with a reasonable computational cost - an accurate estimation of the turbine performance and also quite reliably describe the attended flow-field around the rotor and its wake.

Keywords

Darrieus, wind turbine, unsteady Navier-Stokes simulations, CFD, transitional turbulence model, experiments

Nomenclature

Latin symbols

A	turbine frontal area	[m ²]
ADV	Acoustic Doppler Velocimeter	

41	c	blade chord	[m]
42	c_P	power coefficient	[-]
43	C_A	equivalent chord of the struts	[m]
44	BEM	Blade Element Momentum	
45	CFD	Computational Fluid Dynamics	
46	D	turbine diameter	[m]
47	H	turbine height	[m]
48	k	turbulent kinetic energy	[m ² /s ²]
49	LLT	Lifting Line Theory	
50	O	centroid of the generic element on the strut	
51	RANS	Reynolds Averaged Navier-Stokes	
52	R	turbine radius	[m]
53	Re	Reynolds Number	[-]
54	Re_θ	Momentum Thickness Reynolds Number	[-]
55	S	tunnel cross section area	[m ²]
56	SST	Shear Stress Transport	
57	T	torque per unit length	[Nm/m]
58	TSR	tip-speed ratio	[-]
59	U	wind speed	[m/s]
60	U'	local absolute wind speed on the rotor	[m/s]
61	VAWTs	Vertical-Axis Wind Turbines	
62	W	relative speed	[m/s]
63	y^+	dimensionless wall distance	[-]
64			
65	<u>Greek symbols</u>		
66	γ	intermittency	[-]
67	ϑ	azimuthal angle	[deg]
68	ν	kinematic viscosity	[m ² /s]
69	ρ	density	[kg/m ³]
70	σ	standard deviation of experimental wake data	[-]
71	ω	specific turbulence dissipation rate	[1/s]
72	Ω	revolution speed	[rad/s]
73			
74	<u>Subscripts</u>		
75	0	value at inlet/infinity	
76	ave	averaged value	
77	j	generic element	
78	res	resistant	
79	x	streamwise direction	
80			

81 1. Introduction

82 1.1 Background

83 After most pilot research projects on vertical-axis wind turbines (VAWTs) came to a
84 standstill in the mid 90's [1], these turbines are presently being re-discovered by researchers
85 and manufacturers. Among the different turbine architectures, the Savonius rotor is indeed
86 especially welcome in case of small installations, where an effective self-starting is required
87 [2], in hydrokinetic applications [3] or in hybrid applications as add-on to lift-based devices
88 to improve the self-starting [4]. On the other hand, the Darrieus turbines are suggested as a

89 valuable alternative to horizontal-axis wind turbines (HAWTs) for power production is small
90 and medium-size applications [5] or even in very large ones [6]. Some inherent advantages of
91 the Darrieus concept (performance independence on wind direction, generator positioned on
92 the ground [1], low noise emissions [7], enhanced performance in skewed flows [8], thanks to
93 increased virtual swept area) may indeed outweigh their disadvantages in specific
94 applications. In particular, increasing attention is being devoted to applications in the urban
95 environment [9], where the attended flow conditions include high turbulence levels and
96 misaligned flows, or in floating platforms [10], where the independence on wind direction
97 and the mass concentration at the ground allow a more effective control of the system
98 oscillations. Moreover, VAWTs are often preferred to other turbine types in densely
99 populated areas because they are perceived as aesthetically more pleasant, thus easier to
100 integrate in the landscape [11]. In particular, different architectures are available for this type
101 of rotors [12] in terms of blades' number, blades' shape (straight, helix, bended, etc.) or
102 struts' types.

103 In order to make this technology competitive from an industrial point of view, however,
104 an improvement in the design is needed, particularly focusing on extending the energy
105 harvesting at low wind speeds, as recently shown by [13].

106 The aerodynamic design of these rotors, especially in the "first design phase", has been
107 historically carried out with low-order methods, like the Blade Element Momentum (BEM)
108 theory [14-15] or lifting line theory (LLT) methods [16-17]. More recently, however, the
109 intrinsic limitations of these models, in which the airfoils are modeled with lumped
110 parameters (lift, drag and moment coefficients), made clear to researchers that more powerful
111 tools are needed in order to understand in detail some of the complicated physical phenomena
112 taking place during the revolution of Darrieus rotors. This conclusion was recently drawn in
113 [18], where the authors carried out a critical survey on the accuracy of the aforementioned
114 methods using different test cases. In particular, some of the phenomena that cannot be
115 reduced to a lumped-parameter analysis are the interaction of the blades with macro vortices
116 [19], the flow curvature effects [20-21] (i.e. the effects of virtual airfoil cambering and the
117 extra-incidence induced by the rotation of an airfoil inside a linear flow field) or the dynamic
118 stall [22] induced by the continuous and fast variation of the incidence angle.

119 If experimental testing is often extremely expensive, Computational Fluid Dynamics
120 (CFD) can provide versatile and accurate means to improve the understanding of Darrieus
121 VAWT unsteady aerodynamics and achieve higher-performance in Darrieus turbine design.
122 The use of unsteady Reynolds Averaged Navier-Stokes (RANS) CFD for simulating time-
123 dependent Darrieus turbine aerodynamics is then rapidly increasing due to both the ongoing
124 development and deployment of more powerful high-performance computing hardware (e.g.
125 large clusters of multi- and many-core processors [23]) and also the development of
126 computationally more efficient algorithms or dedicated codes [24].

127 128 *1.2 Literature CFD studies on Darrieus VAWTs*

129 Early assessments of the unsteady RANS CFD technology for Darrieus rotor
130 aerodynamics, aiming primarily at thoroughly investigating the complex fluid mechanics of
131 these machines, made use mainly of a two-dimensional approach (e.g. [25-26]). An extensive
132 literature review on these studies has been recently provided by [27], where the authors
133 presented and explained the most common settings and models used for the simulation of
134 these turbines. Moreover, the authors also proposed a "best simulation" strategy in terms of
135 CFD models, meshing and time-stepping choices, simulation settings.

136 The use of a 2D approach was motivated by the need of containing the computational
137 cost of the simulations themselves, since the fully-unsteady solution of the flow field past
138 rotating Darrieus blades has some strict requirements in terms of both spatial and temporal

139 discretizations. For example, ref. [28] showed that this type of simulation is particularly
140 sensitive to the spatial discretization; in [29], specific criteria based on dimensionless
141 numbers confirmed this assumption and highlighted the need of a remarkable grid refinement
142 in a 1-chord-diameter area around the airfoil. Moreover, according to [29], the Courant
143 number has to be contained to very low values in order to properly capture the transient flow
144 field evolution. If these requirements are fulfilled, recent works (e.g. [30]) showed a
145 promising agreement between 2D simulations and experiments. Notwithstanding this,
146 however, some very interesting phenomena are intrinsically discarded by two-dimensional
147 analyses (e.g. tip effects, vortices propagation in the span direction, etc.).

148 On the other hand, since the very first studies on CFD analyses for Darrieus rotors,
149 researchers have always longed to perform three dimensional tests of these machines in order
150 to fully understand some phenomena that are usually only hypothesized or modeled
151 empirically. In the last few years, thanks to the increase of computational resources, 3D CFD
152 analyses have then received increasing attention and a great number of studies have been
153 published, analysing for example new turbine architectures [32-33], different shapes of the
154 supporting arms [34] or the effect of a misaligned flow [35]. In particular, it was shown in
155 [36] that a 3D approach can definitely improve the accuracy of the simulations and provide a
156 detailed description of the entire flow-field around a H-Darrieus rotor. On the other hand,
157 these simulations become extremely expensive from a computational point of view,
158 especially in case proper spatial and temporal refinement have to be guaranteed, as discussed
159 in [37], where a high-computing cluster was used for a very accurate simulation of a single
160 blade in Darrieus motion. In [36], for example, an increase of the computational cost of about
161 four orders of magnitude between 2D and 3D simulations was noticed. In any case, 3D
162 simulations are presently not affordable for industrial purposes. In this view, the assessment
163 of a 2D strategy able to give sufficiently reliable aerodynamic results is thought of interest,
164 since it would provide an effective tool to achieve a better understanding of some main
165 physical phenomena and thus to ensure a better design of future machines.

166 *1.3 Aim of the present study*

167
168 In the present study, unique experimental data on the performance curve and the near-
169 wake structure of a Darrieus rotor tested in a large tow tank were used as a benchmark to
170 validate the effectiveness of a two-dimensional CFD approach in comparison to a fully three-
171 dimensional one. In particular, the correct definition of the computational domain, the
172 selection of the turbulence model and the correction of simulated data for the parasitic torque
173 components are discussed in this work.

174 The paper is organized as follows: Section 2 summarizes the main features of the case
175 study, the experimental layout and available measurements. Section 3 is dedicated to the
176 description of the numerical approaches, including the analysis of grid characteristics and
177 simulation settings. Results and comparisons with respect to both three-dimensional
178 simulations and experiments are reported in section 4, while a summary of the study and
179 concluding remarks are finally provided in section 5.

180 **2. Case study and experiments**

181 *2.1 Study turbine*

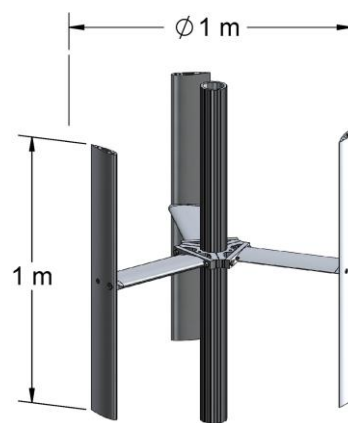
182
183 The turbine modeled in this study (Fig. 1) was the Reference Vertical Axis Turbine
184 (UNH-RVAT) of the University of New Hampshire, developed by the Center for Ocean
185 Renewable Energy (UNH-CORE), in the spirit of, but not geometrically identical to, the DOE

186 Reference Model turbine designed by Sandia National Laboratory in 2011 [38], often referred
187 to as “RVAT” (Reference Vertical Axis Turbine) or “RM2” (Reference Model 2).

188 The model is made of a high solidity (blade chord-to-radius ratio equal to 0.28) rotor
189 constructed from 0.14 m chord length NACA 0020 foils with 1 m span, mounted at 1 m
190 diameter. The blades are supported by three struts attached at mid-chord and mid-span. The
191 struts are also shaped as NACA 0020 airfoils, in order to contain their parasitic torque during
192 the revolution. The central tower was made of an aluminium splined shaft having a mean
193 diameter of 0.095 m. For additional details on the model please refer to [36,39].

194 The UNH-RVAT was the study case of various experimental and numerical
195 investigations [36,39,40], which demonstrated unique near-wake mean velocity field of the
196 rotor, its relevance to wake recovery, and its Reynolds number dependence. In further detail,
197 the open performance and near-wake dataset [41] for the UNH-RVAT was used for
198 validation in the present study.

199



200
201 **Fig. 1 - Study turbine [39].**

202 2.2 Experiments

203 The experimental validation dataset [10] employed here was acquired in the 36 m long
204 UNH-CORE tow tank. Tests were carried out using a water speed of 1 m/s, giving a reference
205 value of the turbine diameter Reynolds number of approximately $1 \cdot 10^6$. More specifically, at
206 the lowest regime (TSR=1.1) the chord-based Reynolds number (calculated with an
207 estimation of the relative speed based on a BEM approach) was approximately in the range
208 between $3.0 \cdot 10^4$ and $3.5 \cdot 10^5$, while at the highest one (TSR=3.2) it was in the range between
209 $4.0 \cdot 10^5$ and $6.8 \cdot 10^5$.

210 The tow tank, whose cross section is sketched in Fig.2, is 2.44 m deep and 3.66 m wide.
211 Based on the present turbine frontal area, the tank cross section produced a blockage ratio of
212 approximately 11%.

213 The experimental setup, including the supporting frame and the instrumentation is also
214 depicted in Fig. 2. Carriage motion was actuated by a permanent magnet servomotor and
215 timing belt, providing highly accurate tow velocities, independently verified by a high-
216 resolution linear encoder. The turbine was installed in the tow tank by means of a dedicated
217 supporting frame, which was built from NACA 0020 struts, mounted to the carriage via linear
218 bearings. The turbine shaft was loaded by a servo motor and gearhead, which provided
219 precise control of mean turbine TSR.

220

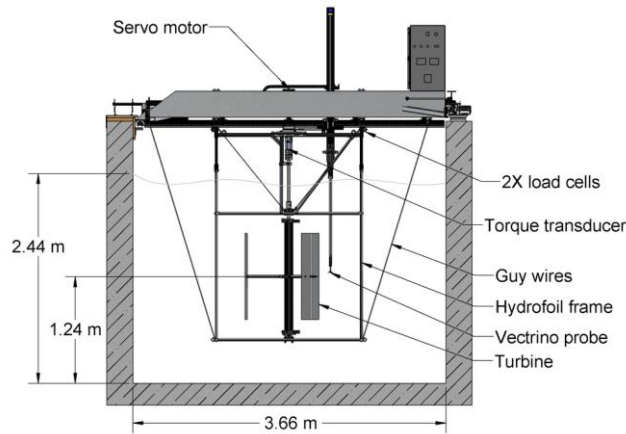


Fig. 2 - Drawing of the turbine test bed installed in the UNH tow tank [39].

Power and overall rotor drag coefficients were measured over a range of TSRs, and the upper half of the wake at TSR=1.9 and one rotor diameter downstream was measured using a Nortek Vectrino+ acoustic Doppler velocimeter (ADV), which has an approximately 6 mm diameter sampling volume and sampled at 200 Hz. The ADV and data acquisition systems' sampling times were synchronized by triggering the start of data acquisition via a pulse sent from the motion controller. Additional details of the turbine and experimental setup are described in [39].

3. Numerical techniques

In this study, different numerical techniques have been considered. In particular, the results of a previous full-3D approach [36] were first considered as a benchmark for other simulations, being the ones thought to provide the most reliable description of the turbine behavior (Section 3.1).

Focus was then given (Section 3.2) to 2D simulations using the commercial solver ANSYS® FLUENT® and the comprehensive approach developed in the last few years by some of the authors [27,29]. In this second type of simulations, dedicated sensitivity analyses on the proper definition of the computational domain (Section 3.3) and on the turbulence model selection (Section 3.4) were also provided.

3.1 Former results of OpenFOAM simulations

The open-source CFD package OpenFOAM has been used recently by one of the authors to model the UNH-RVAT in two and three dimensions using the $\kappa\text{-}\omega$ SST and the Spalart-Allmaras turbulence models [40]. The 2D models both dramatically over predicted performance, whereas the 3D simulations predicted performance more closely matching experiment, with the Spalart-Allmaras model performing best. As will be extensively discussed later on in the study, it should be noted that, in the 2D case simulations, the tow tank width was matched, and therefore blockage was artificially increased, which could explain the exaggerated performance, as more flow was forced through the turbine.

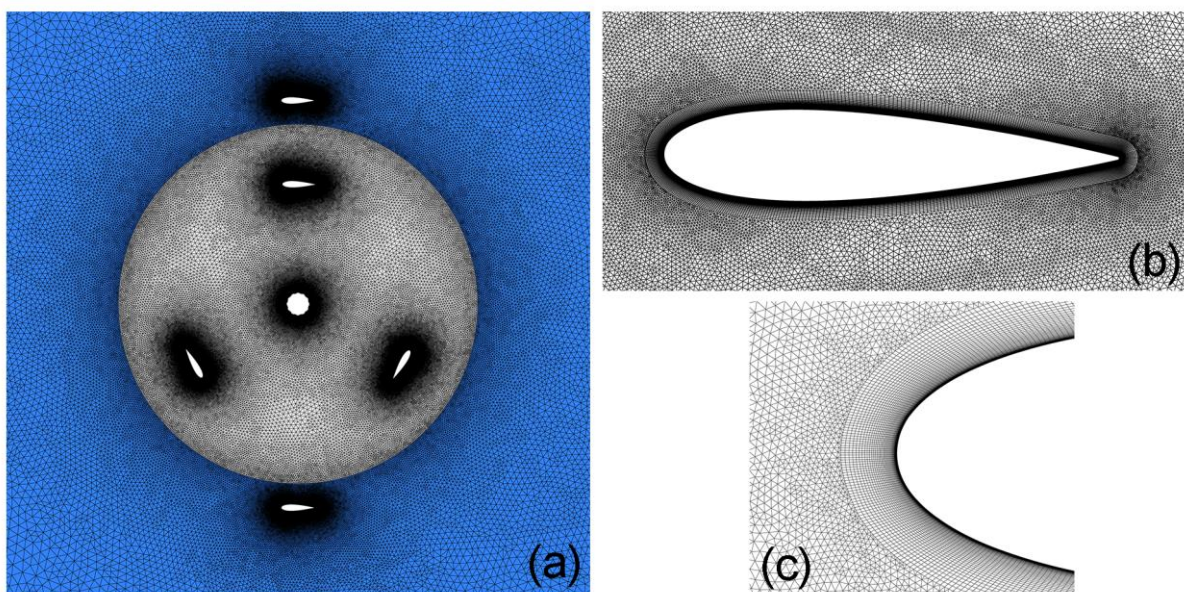
3.2 2D simulations with ANSYS® FLUENT®

A robust numerical approach using the commercial solver ANSYS® FLUENT® [41] has been developed in the last few years by some of the authors (e.g. [27]). The fluid mean was water at exactly the same temperature measured in the experimental tests (19.5 °C), leading to kinematic viscosity $\nu = 1.0115 \cdot 10^{-6} \text{ m}^2/\text{s}$.

256 The time-dependent unsteady RANS approach is used in its pressure based formulation,
257 which showed a higher accuracy for these simulations [27]. The Coupled algorithm is
258 preferred to handle the pressure-velocity coupling, since the dedicated sensitivity analyses of
259 [27] showed a superior robustness of this algorithm when different meshes, timesteps, or
260 rotating speeds are used. The second order upwind scheme is applied to the spatial
261 discretization of the whole set of Navier-Stokes and turbulence equations, as well as the
262 bounded second order for the time differentiation to obtain a good resolution. For each
263 operating condition, the global convergence is defined by fixing a periodicity error threshold
264 equal to 0.1% between the mean values of the torque coefficient over two subsequent
265 revolutions normalized by the mean value over the second period of the pair.

266 In order to allow the physical revolution of the turbine, the sliding-mesh model of the
267 solver is used. The simulation domain, whose dimensions will be discussed in Section 3.3, is
268 then divided into two subdomains [27,43], i.e. a circular rotating zone containing the turbine
269 and a rectangular fixed outer zone determining the overall domain extent (Fig. 3(a)). Further
270 details can be found in Section 3.3.

271 An unstructured triangular mesh is applied to discretize the whole domain, except for
272 the boundary layer region where a quadrilateral structured O-grid is used (Fig. (3b)). In
273 previous studies [27,29,44], it was shown that the number of nodes in which the airfoil is
274 discretized is crucial for the determination of both the attack angle of the incoming flow on
275 the blade and the boundary layer evolution from the leading edge to the trailing edge.
276 Moreover, the discretization level adopted in the near-blade region also strongly impacts on
277 the total number of mesh elements, since the growth of the element size has to be accurately
278 controlled [41].
279



280
281 **Fig. 3 - Some details of the computational mesh: (a) rotating region and supporting**
282 **struts; (b) airfoil; (c) leading edge.**
283

284 For the present study case, initial meshes were created based on previous experience on
285 similar NACA 4-digits VAWTs airfoils [20-21,27,41]; the fulfillment of the thresholds based
286 on the dimensionless numbers presented in [29] was also ensured. A dimensionless wall
287 distance y^+ lower than 1 was guaranteed in all the cells of the domain at any functioning tip-
288 speed ratio. In particular, two different mesh refinements were provided as a function of the
289 TSR for the unstable and the stable halves of the power curve, respectively. This

290 recommendation, proposed by [27] based on detailed comparisons with experimental data, is
 291 based on the fact that in the unstable part of the power curve the airfoils undergo large
 292 variations on the incidence angle which lead to large zones of separated flow around the
 293 airfoils. To properly describe these zones, finer meshes are needed [29], which are conversely
 294 superfluous at higher TSRs.

295 In order to ensure the grid-independency of the results, a first mesh sensitivity analysis
 296 was performed at two TSRs (i.e. the design one of TSR=1.9 and an unstable one, TSR=1.4)
 297 by doubling the number nodes on both the airfoils' surfaces and on a virtual control circle
 298 around them having one chord radius [20-21]. No appreciable torque variation, both in terms
 299 of mean value (always lower than 0.1%) and of torque profiles (coefficient of determination
 300 between the curves higher than 99% [44]), was measured, ensuring the suitability of the
 301 present meshes, whose main features are presented in Tab. 1.

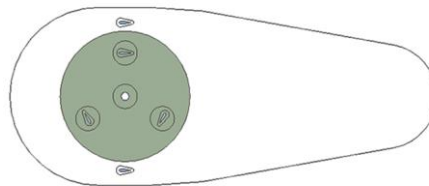
302
303

Tab. 1 - Mesh sensitivity for the rotating domain.

Mesh	TSR range	# elements in the rotating domain	# nodes on the airfoil	# quads' rows in the boundary layer	1 st blade element height
Coarse	1.9 – 3.1	276806	520	50	1 x 10 ⁻⁵ m
Refined	1.0 - 1.4	593880	900	50	1 x 10 ⁻⁵ m

304
305
306
307
308
309
310

In addition to the mesh sensitivity analysis inside the rotating region, which definitely represents the most critical issue in ensuring a proper spatial discretization, a check on the refinement in the turbine wake was also carried out. In particular, a “drop-shaped” control zone (depicted in Fig. 4, together with other refinement zones around the struts and the central tower) was defined during the meshing strategy of the stationary domain.



311
312
313

Fig. 4 - Mesh refinement in the wake.

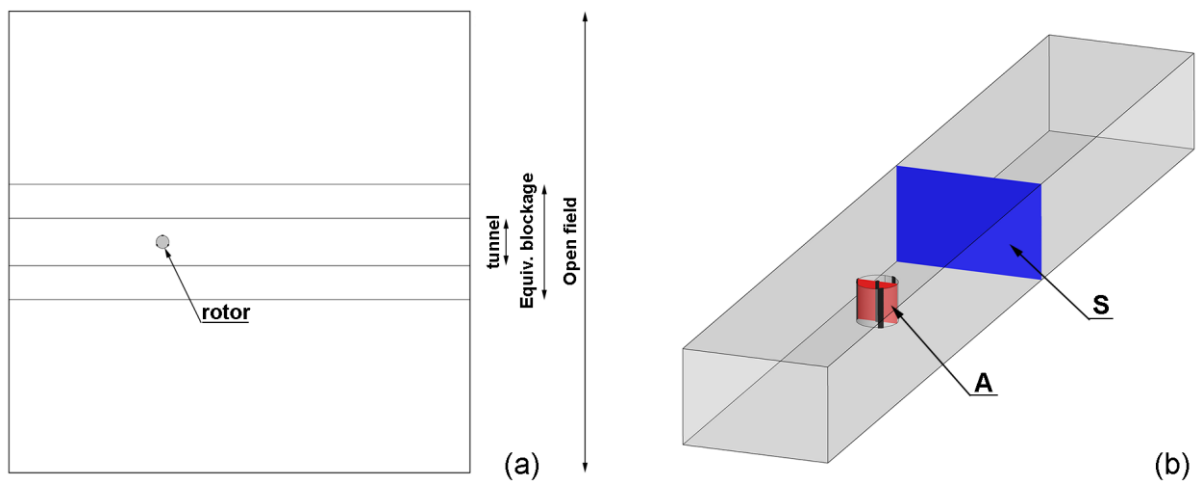
314 In the final version of the stationary mesh (267660 total elements) the elements' size in
 315 the contour of the control drop was selected as to guarantee an expansion ratio from the
 316 sliding interface of approximately 1.1.

317 Starting from this reference dimension, the elements' size was again doubled, obtaining
 318 a null variation of the turbine performance, i.e. mean value difference lower than 0.01%
 319 coefficient of determination between the torque profiles higher than 99,9%.

320 Finally, two different timesteps were also used for the TSRs before or after the curve
 321 peak, respectively. According to [27,29], shorter timesteps must indeed be ensured for the
 322 functioning points in the unstable part of the curve, where large separations and macro-
 323 vortices are expected. Following the general criteria of [29], a timestep of 0.14° was used up
 324 to TSR<1.9 while a timestep of 0.23° was considered for higher TSRs.

325 *3.3 Computational domain sensitivity analysis*

326 As extensively discussed in [27], the dimensions of the computational domain represent
 327 a critical issue for the correct setting of two-dimensional CFD simulations of VAWTs. In the
 328 present study, three different strategies are analyzed and compared (Fig. 5(a)).
 329



330
 331 **Fig. 5 - Computational domain sensitivity analysis: (a) tested distances of lateral**
 332 **boundaries in the 2D case; (b) blockage equivalence between turbine and tunnel frontal**
 333 **areas.**
 334

335 First, the domain dimensions were defined based on the original proportions of the tow
 336 tank in terms of width ($3.7D$) and length ($36D$). This solution, named “tunnel” in the figure,
 337 has been extensively adopted in the literature (e.g. [19,26,36]), even if the final results of the
 338 correspondent simulations were not always in good agreement with experiments.

339 Then, a virtual domain width was defined (named “Equivalent Blockage” in the figure)
 340 able to reproduce the same ratio available in experiments between the area of the rotor A and
 341 the tunnel area S (see Fig. 5(b)). This solution was generated based on the general criteria
 342 used to transpose boundary conditions from a three-dimensional domain to a two-
 343 dimensional one, in which flow passage areas become lines. In this configuration, the width
 344 was approximately $9D$.

345 Finally, an “Open field” like configuration was considered ($36D$), in which the domain
 346 lateral boundaries were placed far enough to avoid any influence on the rotor, following the
 347 sensitivity analysis presented in [27]. This configuration, conventionally used by the authors
 348 in recent works [20-21,27,29], ensured a good level of accuracy of numerical results when
 349 compared to experiments, even if acquired in a confined wind tunnel. The use of this
 350 condition is even more relevant for the present case study since the tow tank is a free surface
 351 channel, thus ensuring an even lower actual blockage thanks to the variation of the water
 352 level.

353 *3.4 Turbulence models*

354 The turbulence closure using the $k-\omega$ SST model by Menter [45] has been shown in the
 355 last few years to be promisingly effective in the CFD study of Darrieus wind turbines, if
 356 compared to other models, as discussed in [30]. Some of the authors recently showed that it is
 357 able to ensure a better matching with experiments in the correct location of stall on the
 358 airfoils during the revolution [27]. Simulations using the approach described in Section 3.2
 359 and the $k-\omega$ SST turbulence model were also generally able to properly describe both the
 360 power curve of the experimental rotors (e.g. see [27,45]).

361 As discussed in [20-21], however, whenever the attended blade Reynolds numbers are
 362 low enough to make one expect transitional effects (e.g. small rotors at low tip-speed ratios),
 363 the use of a transitional turbulence model can be suggested. In particular, the $\gamma-Re_{\theta}$ transition
 364 model (derived from the SST model [47]) was successfully implemented in [20-21], despite
 365 its increased computational cost. In addition, good agreement between experimental data and
 366 numerical results was obtained in [48] with the transition turbulence model for two different
 367 types of H-Darrieus turbines.

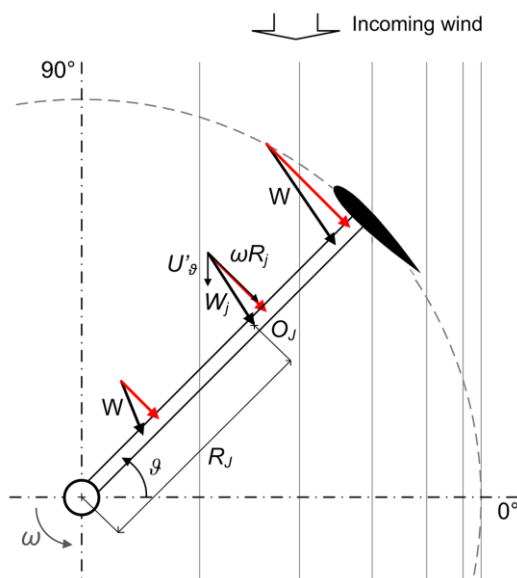
368 On these bases, both turbulence models were used and compared in the present study,
 369 showing the additional benefits of the turbulence model especially at the low TSRs of the
 370 considered rotor.

371 3.5 Resistant torque correction

372 As discussed in [40], experiments have been corrected for the parasitic torque coming
 373 from the bearings' friction. However, no correction for the aerodynamic resistant torque has
 374 been originally provided. In order to make CFD data fully comparable to the experiments, the
 375 attended resistant torque was then subtracted to purely aerodynamic data.

376 The resistant torque was estimated using the lumped parameters model developed by
 377 the University of Firenze in [49]. The model makes use of the BEM multitubes approach for
 378 the discretization of the rotor; the normal component of the relative velocity on the struts W_{\perp}
 379 (i.e. the one really producing drag [50]) is then punctually evaluated at a discrete number of
 380 positions O_j as a function of the wind velocity reduced by the induction factor of the BEM
 381 model U'_{ϑ} , the azimuthal angle and the local radius R_j for each rotational speed Ω (see Fig.
 382 6). The equivalent drag coefficient of the struts is given as a function of both the struts cross
 383 section and the relative Reynolds number. Based on these hypotheses, the average parasitic
 384 torque of a rotating strut at a given TSR (i.e. a given Ω) is given by Eq. 1, where C_A is the
 385 equivalent chord of the strut.

386



387

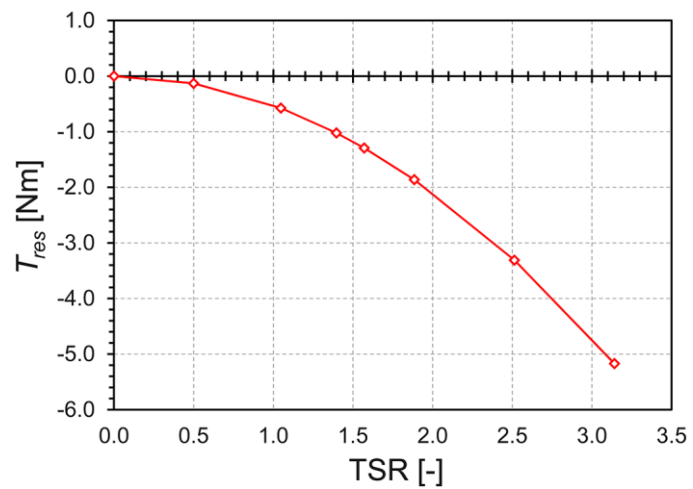
388 **Fig. 6 - Model for the calculation of the struts' resistant torque.**

389

$$390 \quad T_{res_ave}(\Omega) = \frac{1}{4\pi} \rho C_A \int_0^{2\pi} \int_{D_r/2}^R C_D(\Omega, \vartheta, R) \cdot W_{\perp}(\vartheta, R)^2 dR d\vartheta \quad (1)$$

391

392 The model has been validated with several experimental campaigns (e.g. [49,51]),
 393 obtaining a satisfactory agreement. Based on the model, the resistant torque of the present
 394 study turbine as a function of TSR is reported in Fig. 7.
 395

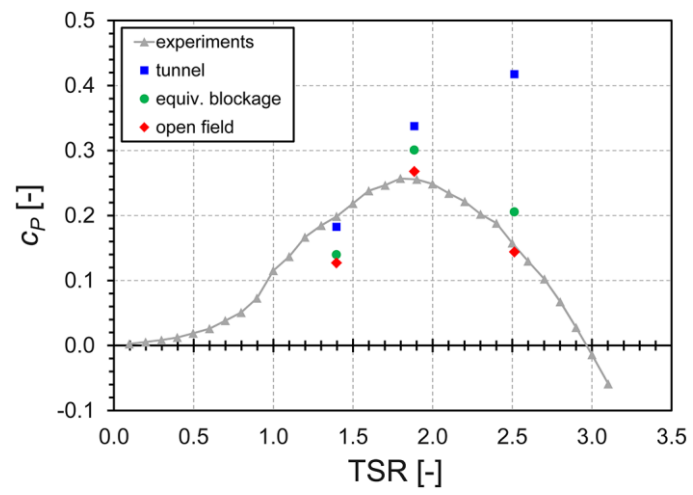


396 **Fig. 7 - Calculated resistant torque for the present study turbine.**

396
 397
 398
 399 **4. Results**

400 *4.1 Sensitivity analysis on lateral boundaries*

401 The sensitivity analysis on the domain size was first carried out. Three relevant tip-
 402 speed ratios were considered for this analysis, namely TSR=1.4, TSR=1.9 and TSR=2.5. The
 403 first one lies in the unstable part of the operating curve, the second is located near the peak
 404 performance, while the third represents a stable functioning condition. Fig. 8 compares the
 405 experimental power curve of the rotor with the numerical results using the “tunnel”, the
 406 “equivalent blockage” and the “open field” configurations described in Section 3.3 (all
 407 corrected for the resistant torque).
 408



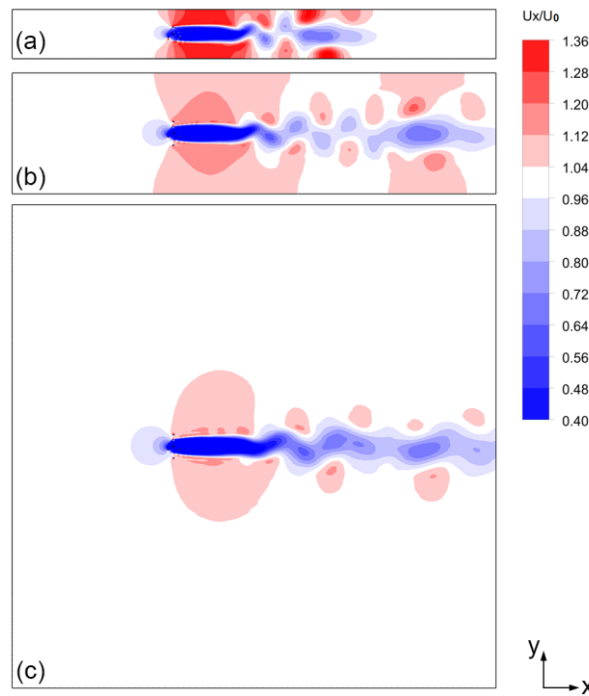
409 **Fig. 8 - Comparison between experiments and simulations using different strategies in**
 410 **the selection of the later boundaries.**
 411
 412

413 Upon examination of the figure, it is apparent that results using the “tunnel”
 414 configuration are notably far from the experimental trend, with a completely wrong
 415 prediction of the curve shape and of the modulus of maximum power coefficient. These

416 results are indeed in agreement with the former reported in [40] made with the same domain
417 dimensions with the OpenFOAM CFD package and the same turbulence model.

418 When the lateral boundaries are distanced, the agreement with experiments notably
419 increases, reaching its maximum for the “open field” configuration, which is able to properly
420 describe the shape of the curve and accurately predict the two stable power coefficient values.
421 To further stress the differences induced by the boundaries on the computed flow field, Fig. 9
422 compares the CFD results at TSR=3.3, where the difference is larger, in terms of velocity
423 component in the streamwise direction normalized by that at the inlet, i.e. U_x/U_{x0} .

424



425

426

Fig. 9 - Normalized velocity component in the streamwise direction (U_x/U_{x0}) for three simulated domains at TSR=2.5.

427

428

429

430

431

432

433

434

435

436

437

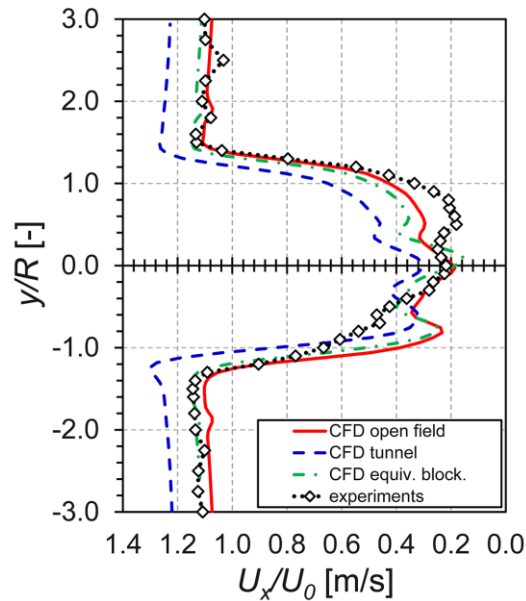
438

439

440

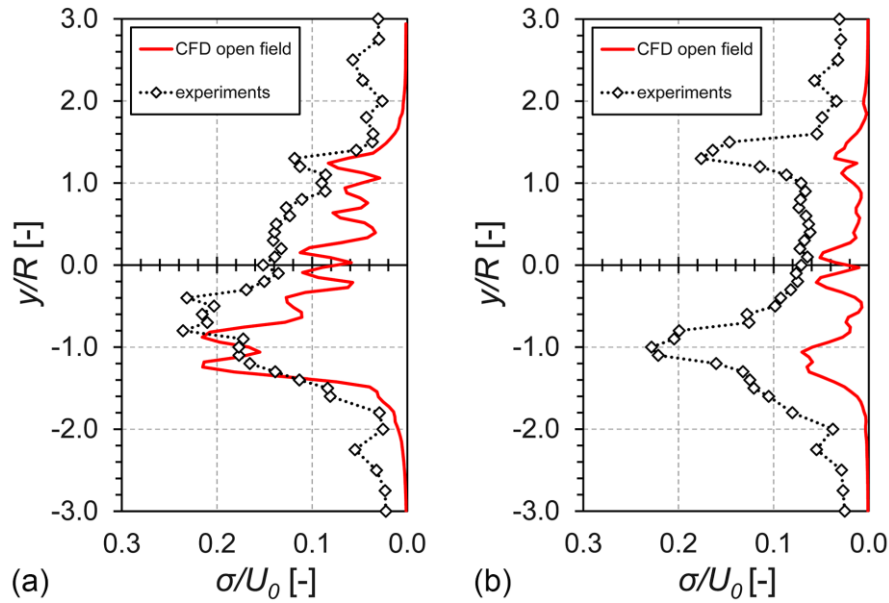
441

Upon examination of the figure, it is apparent that the lateral boundaries impose a strong acceleration to the flow when approaching the rotor. The turbine wake is also contracted and decays faster than in the “open field” configuration. Overall, the entire shape of the flow field around turbine is strongly modified: the turbine performance is consequently much different, as testified by Fig. 8. If the superior accuracy of 2D simulations using the “open field” configuration in predicting the performance curve was already discussed by Balduzzi et al. [27], the possibility of exploiting the unique wake measurements provided by the present experiments allowed to verify the reasons of this evidence. To do so, Fig. 10 compares the wake measurements at TSR=1.9 (similar results were found for TSR=1.4 but not reported here for brevity) with the numerical results extracted from the computed flow field in a virtual rake positioned exactly in the same position than the experimental one; CFD data were averaged on 5 revolutions.



442
 443 **Fig. 10 - Normalized velocity profiles in the wake at TSR=1.9: comparison between**
 444 **experiments and simulations using different lateral boundaries.**
 445

446 Upon examination of the figure, it has to be noted that some differences are present
 447 between numerical simulations and experiments, especially regarding the non-symmetric
 448 nature of the wake. On the other hand, the comparison readily shows that the simulations
 449 using the lateral boundaries positioned at a distance equal to that of tunnel blades were also
 450 not able to properly predict the velocity levels both inside the wake and in its proximity, with
 451 a general overestimation in comparison to experiments. In fact, since in 2D simulations the
 452 flow cannot have velocity components along the blade span, the entire mass flow is indeed
 453 forced to pass through the rotor plane: by doing so, a “virtual” additional blockage is
 454 produced, as clearly demonstrated by the strong acceleration predicted by these simulations
 455 around the rotor (i.e. outside $-1 < y/R < 1$). In addition, the velocity deficit in the wake is also
 456 underestimated, leading to the excessive power production showed by Fig. 8. Both the
 457 “equivalent blockage” and the “open field” simulations, conversely, properly predicted the
 458 velocity levels almost everywhere, with the “open field” approach being able to more closely
 459 match the peak of velocity deficit downstream the rotor and the overall shape of the wake.
 460 The accuracy of these latter CFD simulations was also testified by the comparison of
 461 normalized standard deviations for the wake measurements, reported in Fig. 11 for both TSRs
 462 available from the experimental campaign.
 463



464
 465 **Fig. 11 - Normalized standard deviation of velocity measurements in the wake at**
 466 **TSR=1.4 (a) and TSR=1.9 (b): comparison between experiments and simulations using**
 467 **the “open field” configuration.**
 468

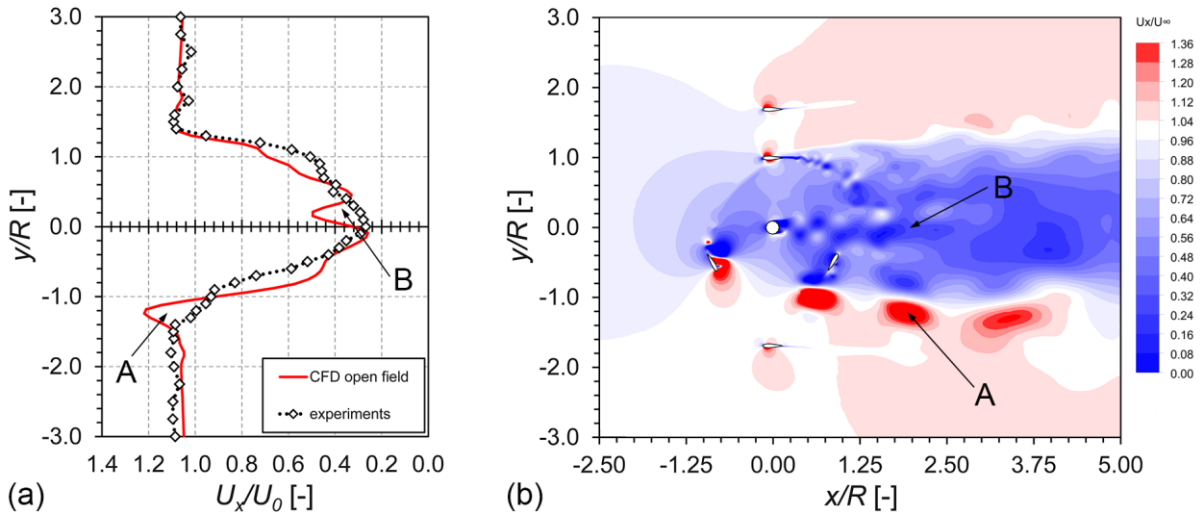
469 As expectable, CFD generally predicted lower values of the measurements’ standard
 470 deviation with respect to experiments. Notwithstanding this, the distribution of standard
 471 deviation was sufficiently well reproduced, suggesting that the main aerodynamic phenomena
 472 inducing an increase of the measurements’ scatter (e.g. vortices or separated flows) were
 473 captured. According to the results of Fig. 10, a discrepancy is noticed in the wake just behind
 474 the tower, where CFD generally tends to predicted more intense ripples with respect to
 475 experiments.

476 Based on the results of this first set of simulations, it was concluded that the “open
 477 field” configuration can be considered the most effective way of simulating Darrieus VAWTs
 478 whenever only two-dimensional simulations are available. In this view, this approach is
 479 assumed to provide valuable data for comparison even with experiments carried out in a
 480 tunnel (properly corrected for blockage, whenever needed).

481 *4.2 Detailed wake analysis*

482 The “open field” simulations at TSR=1.4 and TSR=1.9 were then further exploited to
 483 carry out detailed wake analyses, since for both functioning points experimental
 484 measurements were available from [39].

485 Figure 12 first compares the velocity profiles in the wake (a) and displays the
 486 normalized velocity field (b) in the streamwise direction predicted by CFD at TSR=1.4.
 487



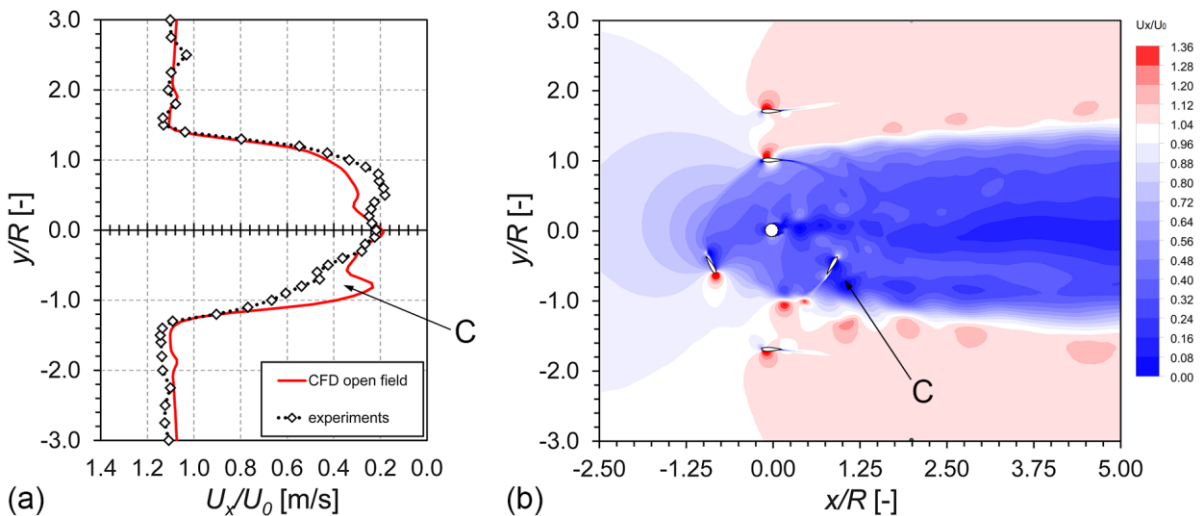
(a) (b)

Fig. 12 - Normalized velocity in the wake at TSR=1.4: comparison between experiments and simulations (a) and computed velocity field (b).

488
489
490
491
492
493
494
495
496
497
498
499
500
501
502
503
504
505
506
507
508

Upon examination of Fig. 12(a), sound agreement can be noticed between experiments and simulations, even if these latter ones underestimated the wake deflection in the counterclockwise direction (i.e. the wake is more symmetric with respect to the center of the machine). Moreover, two main ripples are present in CFD simulations, marked as phenomenon “A” and phenomenon “B” in the figure. Even though it must be remembered that Fig. 12(a) indeed represents an averaged profile while Fig. 12(b) is a snapshot of the flow field at a given instant in time, it is readily arguable that the ripple A in CFD is connected to the strong vortices detached upwind from the blades and then convected by the flow downstream, whose intensity was probably overestimated by numerical simulations with respect to experiments. On the other hand, phenomenon B is associated to an excessive velocity deficit in the wake of the tower, again not seen in experiments. Overall, it can be noticed that a superior uniformity was noticed in experiments with respect to simulations, probably due also the additional mixing in the third dimension which is not allowed by the 2D simulations.

Moving to TSR=1.9, Fig. 13 again compares the velocity profiles in the wake (a) and displays the normalized velocity field (b) in the streamwise direction predicted by CFD.



(a) (b)

Fig. 13 - Normalized velocity in the wake at TSR=1.9: comparison between experiments and simulations (a) and computed velocity field (b).

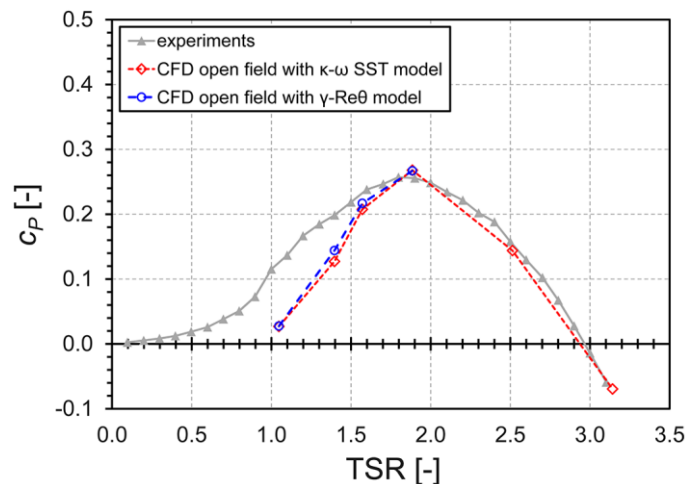
509
510
511

512
513
514
515
516
517
518
519

The overall agreement is very good also at this TSR, with again only a slight underestimation of the wake deflection by CFD. A discrepancy (phenomenon “C”) was also found at approximately $y/R=-0.8$, where simulations predict a wake with very low speed resulting from massive separations in the suction side of the blades entering the downwind region of the rotor. A small trace of such wake is visible also in experiments even though, as already noticed for $TSR=1.4$, a smoother profile and a higher mixing was measured in experiments.

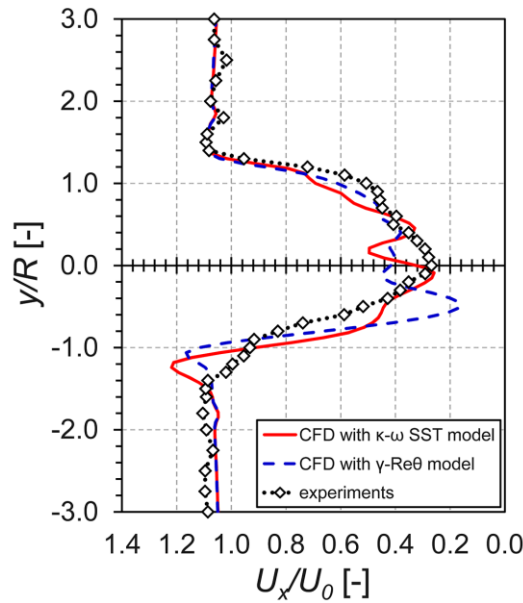
520 4.2 Benefits of a transitional turbulence model at low tip-speed ratios

521 As discussed in Section 3.4, whenever the attended blade Reynolds numbers are low
522 enough to make one expect transitional effects (e.g. small rotors at low tip-speed ratios), the
523 use of a transitional turbulence model has been suggested since it is thought to provide
524 possible benefits in terms of simulations’ accuracy [20-21]. On this basis, the functioning
525 points in the unstable part of the performance curve of the rotor were simulated using both
526 the $\kappa-\omega$ SST model and the $\gamma-Re_\theta$ model. An additional test at $TSR=2.5$ revealed that the
527 change in the turbulence model did not induce any modification in the predicted torque
528 profile at high TSRs. Figure 14 reports the comparison between the two sets of simulations.
529



530
531 **Fig. 14 - Comparison between experiments and simulations using either the $\kappa-\omega$ SST**
532 **model or the $\gamma-Re_\theta$ model at low TSRs.**
533

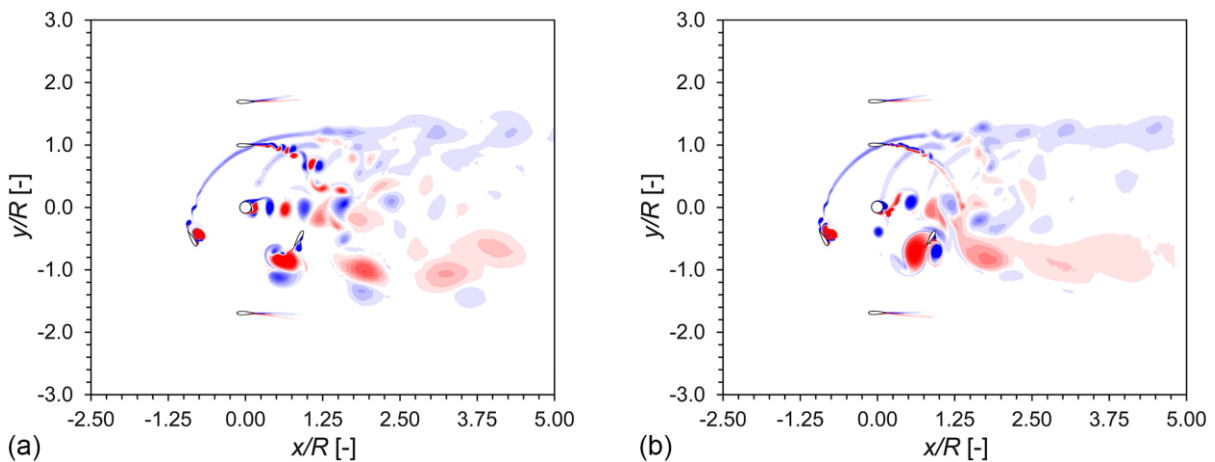
534 Upon examination of the figure, it is apparent that both models actually give coherent
535 results, with the transitional model ensuring constantly a slightly higher performance (i.e. a
536 little bit more similar to experiments) than the conventional SST model. However, none of
537 the two models was able to take simulations substantially closer to experiments. This is
538 probably due to the fact that in this region of the operating curve, the airfoils work in stalled
539 conditions for a considerable fraction of the revolution, with severe flow separation
540 phenomena and large-scale vortices detaching even from the leading edges of the airfoils
541 (already discussed in the previous section). In these conditions, additional elements like the
542 surface roughness, the trailing-edge refinement in the experimental model, etc. can play a
543 fundamental role in setting the characteristics of such separations, which are however pivotal
544 for the final torque production. More interesting information can be however obtained when
545 again looking at the wake analysis. In particular, Fig. 15 reports the comparison between
546 experiments and simulations at $TSR=1.4$.
547



548
549 **Fig. 15 - Normalized velocity in the wake at TSR=1.4: comparison between experiments**
550 **and simulations using either the $\kappa\text{-}\omega$ SST model or the $\gamma\text{-}Re_\theta$ model.**
551

552 As one may notice, the use of the transitional turbulence model notably modified the
553 description of the flow field around the turbine. The wake deflection is now a little more
554 accurately described (see particularly the range between $0.8 < y/R < 1.4$) as well as the wake of
555 the tower, were the fluctuations are now reduced. The same behaviour was noticed in all the
556 four investigated TSRs. On the other hand an increased velocity deficit was noticed around
557 $y/R = -0.5$, not present in experiments.

558 To further stress the impact of the different turbulence model, Fig. 16 reports the
559 comparison of predicted vorticity contours at TSR=1.4 using either the $\kappa\text{-}\omega$ SST model (a) or
560 the $\gamma\text{-}Re_\theta$ model (b).
561



562 (a) (b)
563 **Fig. 16 - Vorticity contours at TSR=1.4 using either the $\kappa\text{-}\omega$ SST model (a) or the $\gamma\text{-}Re_\theta$**
564 **model (b).**
565

566 Upon examination of the figure, it is apparent that remarkable differences are
567 introduced in the flow field description by a different handling of turbulence closure. In
568 particular, as expected based on the analysis of Fig. 15, the use of the transitional model leads
569 to a different resolution of the propagation of stall vortices detaching from the blades at the
570 end of the upwind sector. Moreover, the shedding taking place around the tower is also

571 differently solved, avoiding the too strong ripples that were not seen in experiments. Overall,
572 the transitional model apparently leads to a higher mixing of flow structures (vortices and
573 wakes), more similar to what expected based on the experimental measurements. As a final
574 remark, the $\gamma-Re_{\theta}$ model was thought to provide a more accurate resolution of the flow past
575 the blades and its use is then suggested whenever transitional effects may play a role in the
576 airfoils' behavior.

577

578 **5. Conclusions**

579 In the study, unique experimental results collected in a water tow tank on the
580 performance and wake characteristics of a H-Darrieus wind turbine were used to assess the
581 effectiveness of a two-dimensional CFD U-RANS simulation approach.

582 Upon examination of cross-comparisons between experiments and simulations, it was
583 apparent that, even though some simplifications are about to be introduced, a 2D simulation
584 can provide quite accurate estimations of both the overall performance and the flow field
585 description around the rotor with reasonable computational cost, on condition that proper
586 settings are applied. In particular, it is here suggested that, in case of a two-dimensional
587 simulation, the lateral boundaries of the computational domain must be placed sufficiently far
588 from the rotor, in order to have an "open-field-like" configuration; by doing so, any artificial
589 blockage due to the absence of the third dimension is avoided.

590 Moreover, the results further confirmed that, in case of medium-size rotor and low tip-
591 speed ratios, the use of a transitional model for turbulence closure is suggested, since it
592 ensures a more accurate description of the flow transition over the airfoils and of the vortices
593 detachment for incidence angles higher than the stall one.

594

595 **Acknowledgements**

596 The authors would like to acknowledge Prof. Ennio Antonio Carnevale of the
597 Università degli Studi di Firenze for supporting this research activity. Thanks are also due to
598 Dr. Giacomo Benassai for his contribution to the simulations during his MSc thesis at the
599 Università degli Studi di Firenze.

600

601 **References**

602 [1] Paraschivoiu I. Wind turbine design with emphasis on Darrieus concept. Polytechnic
603 International Press: Montreal (Canada), 2002.

604 [2] Altan BD, Atilgan M, An experimental and numerical study on the improvement of the
605 performance of Savonius wind rotor. Energy Conversion and Management
606 2008;49(12):3425-3432.

607 [3] Sarma NK, Biswas A, Misra RD. Experimental and computational evaluation of
608 Savonius hydrokinetic turbine for low velocity condition with comparison to Savonius
609 wind turbine at the same input power. Energy Conversion and Management
610 2014;83:88-98.

611 [4] Bhuyan S, Biswas A. Investigation on self-starting and performance characteristics of
612 simple h and hybrid H-Savonius vertical axis wind rotors. Energy Conversion and
613 Management 2014;87:859-867.

614 [5] Tjiu W, Marnoto T, Mat S, Ruslan MH, Sopian K. Darrieus vertical axis wind turbine
615 for power generation I: Assessment of Darrieus VAWT configurations. Renewable
616 Energy 2015; 75(March 2015): 50-67. DOI: 10.1016/j.renene.2014.09.038

- 617 [6] Tjiu W, Marnoto T, Mat S, Ruslan MH, Sopian K. Darrieus vertical axis wind turbine
618 for power generation II: Challenges in HAWT and the opportunity of multi-megawatt
619 Darrieus VAWT development. *Renewable Energy* 2015; 75(March 2015):560-571.
620 DOI: 10.1016/j.renene.2014.10.039
- 621 [7] Mohamed MH. Aero-acoustics noise evaluation of H-rotor Darrieus wind turbines.
622 *Energy* 2014; 65(1): 596-604. DOI: 10.1016/j.energy.2013.11.031.
- 623 [8] Bianchini A, Ferrara G, Ferrari L, Magnani S. An improved model for the performance
624 estimation of an H-Darrieus wind turbine in skewed flow. *Wind Engineering* 2012;
625 36(6): 667-686. DOI: 10.1260/0309-524X.36.6.667
- 626 [9] Balduzzi F, Bianchini A, Carnevale EA, Ferrari L, Magnani S. Feasibility analysis of a
627 Darrieus vertical-axis wind turbine installation in the rooftop of a building. *Applied*
628 *Energy* 2012; 97: 921–929. DOI: 10.1016/j.apenergy.2011.12.008
- 629 [10] Borg M, Collu M, Brennan FP. Offshore floating vertical axis wind turbines:
630 advantages, disadvantages, and dynamics modelling state of the art. *Marine & Offshore*
631 *Renewable Energy Congress*, London (UK), 26-27 September, 2012.
- 632 [11] Mertens S. *Wind Energy in the Built Environment*. Multi-Science: Brentwood (UK),
633 2006.
- 634 [12] Aslam Bhutta MM, Hayat N, Farooq AU, Ali Z, Jamil ShR, Hussain Z, Vertical axis
635 wind turbine – A review of various configurations and design techniques. *Renewable*
636 *and Sustainable Energy Reviews* 2012;16(4):1926-1939.
- 637 [13] Bianchini A, Ferrara G, Ferrari L. Design guidelines for H-Darrieus wind turbines:
638 Optimization of the annual energy yield. *Energy Conversion and Management*
639 2015;89:690-707. DOI: 10.1016/j.enconman.2014.10.038
- 640 [14] Brahim M, Allet A, Paraschivoiu I. Aerodynamic analysis models for vertical-axis
641 wind turbines. *International Journal of Rotating Machinery* 1995; 2(1): 15-21. DOI:
642 10.1155/S1023621X95000169
- 643 [15] Bianchini A, Ferrari L, Carnevale EA. A model to account for the Virtual Camber
644 Effect in the Performance Prediction of an H-Darrieus VAWT Using the Momentum
645 Models. *Wind Engineering* 2011;35(4):465-482. DOI: 10.1260/0309-524X.35.4.465
- 646 [16] Marten D, Bianchini A, Pechlivanoglou G, Balduzzi F, Nayeri CN, Ferrara G,
647 Paschereit CO, Ferrari L. Effects of airfoil's polar data in the stall region on the
648 estimation of Darrieus wind turbines performance. *Proc. of the ASME Turbo Expo*
649 2016, Seoul, South Korea, June 13-17, 2016.
- 650 [17] Marten D, Lennie M, Pechlivanoglou G, Nayeri CD, Paschereit CO. Implementation,
651 Optimization and Validation of a Nonlinear Lifting Line Free Vortex Wake Module
652 within the Wind Turbine Simulation Code QBlade. *Proc. of the ASME Turbo Expo*
653 2015, Montréal, Canada, June 15-19, 2015.
- 654 [18] Deglaire P. *Analytical Aerodynamic Simulation Tools for Vertical Axis Wind*
655 *Turbines*. Digital Comprehensive Summaries of Uppsala Dissertations from the Faculty
656 of Science and Technology 2010, 704, ISSN 1651-6214.
- 657 [19] Amet E, Maitre T, Pellone C, Achard JL. 2D Numerical Simulations of Blade-Vortex
658 Interaction in a Darrieus Turbine. *Journal of Fluids Engineering* 2009; 131: 111103.1–
659 111103.15. DOI: 10.1115/1.4000258

- 660 [20] Rainbird J, Bianchini A, Balduzzi F, Peiro J, Graham JMR, Ferrara G, Ferrari L. On the
661 Influence of Virtual Camber Effect on Airfoil Polars for Use in Simulations of Darrieus
662 Wind Turbines. *Energy Conversion and Management* 2015;106:373-384. DOI:
663 10.1016/j.enconman.2015.09.053
- 664 [21] Bianchini A, Balduzzi F, Ferrara G, Ferrari L. Virtual incidence effect on rotating
665 airfoils in Darrieus wind turbines. *Energy Conversion and Management* 2016; 111(1
666 March 2016): 329-338. DOI: 10.1016/j.enconman.2015.12.056
- 667 [22] Simao-Ferreira C, van Zuijlen A, Bijl H, van Bussel G, van Kuik G. Simulating
668 dynamic stall on a two-dimensional vertical-axis wind turbine: verification and
669 validation with particle image velocimetry data. *Wind Energy* 2010; 13: 1-17. DOI:
670 10.1002/we.330
- 671 [23] Salvadore F, Bernardini M, Botti M. GPU accelerated flow solver for direct numerical
672 simulation of turbulent flows. *Journal of Computational Physics* 2013; 235: 129-142.
673 DOI: 10.1016/j.jcp.2012.10.012
- 674 [24] Balduzzi F, Bianchini A, Gigante FA, Ferrara G, Campobasso MS, Ferrari L.
675 Parametric and Comparative Assessment of Navier-Stokes CFD Methodologies for
676 Darrieus Wind Turbine Performance Analysis. *Proc. of the ASME Turbo Expo 2015,*
677 *Montreal, Canada, June 15-19, 2015.* DOI: 10.1115/GT2015-42663
- 678 [25] Howell R, Qin N, Edwards J, Durrani N. Wind tunnel and numerical study of a small
679 vertical axis wind turbine. *Renewable Energy* 2010; 35: 412-422. DOI:
680 10.1016/j.renene.2009.07.025
- 681 [26] Raciti Castelli M, Englaro A, Benini E. The Darrieus wind turbine: Proposal for a new
682 performance prediction model based on CFD. *Energy* 2011; 36: 4919-4934. DOI:
683 10.1016/j.energy.2011.05.036
- 684 [27] Balduzzi F, Bianchini A, Maleci R, Ferrara G, Ferrari L. Critical issues in the CFD
685 simulation of Darrieus wind turbines. *Renewable Energy* 2016; **85**(01): 419-435. DOI:
686 10.1016/j.renene.2015.06.048
- 687 [28] Almohammadi KM, Ingham DB, Ma L, Pourkashan M. Computational fluid dynamics
688 (CFD) mesh independency techniques for a straight blade vertical axis wind turbine.
689 *Energy* 2013; 58(1 September 2013): 483-493. DOI: 10.1016/j.energy.2013.06.012
- 690 [29] Balduzzi F, Bianchini A, Ferrara G, Ferrari L. Dimensionless numbers for the
691 assessment of mesh and timestep requirements in CFD simulations of Darrieus wind
692 turbines. *Energy* 2016; **97**(15 February 2016): 246-261. DOI:
693 10.1016/j.energy.2015.12.111
- 694 [30] Daróczy L, Janiga G, Petrasch K, Webner M, Thévenin D. Comparative analysis of
695 turbulence models for the aerodynamic simulation of H-Darrieus rotors. *Energy* 2015;
696 90(1 October 2015): 680-690. DOI: 10.1016/j.energy.2015.07.102
- 697 [31] Lam HF, Peng HY. Study of wake characteristics of a vertical axis wind turbine by
698 two- and three-dimensional computational fluid dynamics simulations. *Renewable*
699 *Energy* 2016; 90(May 2016): 386-398. DOI: 10.1016/j.renene.2016.01.011
- 700 [32] Ghosh A, Biswas A, Sharma KK, Gupta R. Computational analysis of flow physics of a
701 combined three bladed Darrieus Savonius wind rotor. *Journal of the Energy Institute*
702 2015; 88(4): 425-437. DOI: 10.1016/j.joei.2014.11.001

- 703 [33] Alaimo A, Esposito A, Messineo A, Orlando C, Tumino D. 3D CFD Analysis of a
704 Vertical Axis Wind Turbine. *Energies* 2015; 8: 3013-3033. DOI: 10.3390/en8043013
- 705 [34] De Marco A, Coiro DP, Cucco D, Nicolosi F. A Numerical Study on a Vertical-Axis
706 Wind Turbine with Inclined Arms. *International Journal of Aerospace Engineering*
707 2014; 2014: 1-14. DOI: 10.1155/2014/180498
- 708 [35] Orlandi A, Collu M, Zanforlin S, Shires A. 3D URANS analysis of a vertical axis wind
709 turbine in skewed flows. *Journal of Wind Engineering and Industrial Aerodynamics*
710 2015; 147(December 2015): 77-84. DOI: 10.1016/j.jweia.2015.09.010
- 711 [36] Bachant P, Wosnik M. Modeling the near-wake of a vertical-axis cross-flow turbine
712 with 2-D and 3-D RANS. *Journal of Renewable and Sustainable Energy*
713 2016;8(5):053311-1-10. DOI: 10.1063/1.4966161
- 714 [37] Balduzzi F, Drofelnik J, Ferrara G, Ferrari L, Campobasso MS. Darrieus Wind Turbine
715 Blade Unsteady Aerodynamics: a Three-Dimensional Navier-Stokes CFD assessment.
716 Paper submitted to: *Energy*.
- 717 [38] Barone M, Griffith T, Berg J. Reference model 2: rev 0 rotor design. Tech. Rep.
718 SAND2011-9306, Sandia National Laboratories, November 2011.
- 719 [39] Bachant, P, Wosnik, M. Characterising the near-wake of a cross-flow turbine. *Journal*
720 *of Turbulence* 2015; 16(4):392–410. DOI:10.1080/14685248.2014.1001852
- 721 [40] Bachant, P, Wosnik, M. Effects of Reynolds number on the energy conversion and
722 near-wake dynamics of a high solidity vertical-axis cross-flow turbine. *Energies*
723 2016;9(2):73/1-18. DOI:10.3390/en9020073
- 724 [41] Bachant, P, Wosnik, M. UNH-RVAT baseline performance and near-wake
725 measurements: Reduced dataset and processing code. Online tech. rep. DOI:
726 10.6084/m9.figshare.1080781
- 727 [42] Ansys, Inc., 2015, *Fluent Theory Guide*, release 16.1.
- 728 [43] Maître T, Amet E, Pellone C, Modeling of the Flow in a Darrieus Water Turbine: Wall
729 Grid Refinement Analysis and Comparison with Experiments. *Renewable Energy*
730 2013;51:497–512. DOI: 10.1016/j.renene.2012.09.030
- 731 [44] Balduzzi F, Bianchini A, Maleci R, Ferrara G, Ferrari L. Blade design criteria to
732 compensate the flow curvature effects in H-Darrieus wind turbines. *Journal of*
733 *Turbomachinery* 2015;137(1):1-10. DOI: 10.1115/1.4028245
- 734 [45] Menter FR, Two-Equation Eddy-Viscosity Turbulence Models for Engineering
735 Applications. *AIAA J.* 1994;32(8):1598–1605.
- 736 [46] Gigante FA, Balduzzi F, Bianchini A, Yan M, Ferrara G, Campobasso MS, Ferrari L.
737 On the Application of the Reynolds-Averaged Navier-Stokes Equations and the Shear
738 Stress Transport Turbulence Model for the Performance Estimation of Darrieus Wind
739 Turbines. Paper submitted to: *Journal of Wind Engineering and Industrial*
740 *Aerodynamics*.
- 741 [47] Langtry RB, Menter FR, Correlation-based transition modeling for unstructured
742 parallelized computational fluid dynamics codes. *AIAA Journal* 2009;47(12):2894–
743 2906.
- 744 [48] Lanzafame R, Mauro S, Messina M, 2D CFD modeling of H-Darrieus wind turbines
745 using a transition turbulence model. *Energy Procedia* 2014;45:131-140.

- 746 [49] Bianchini A, Ferrari L, Magnani S, Start-up behavior of a three-bladed H-Darrieus
747 VAWT: experimental and numerical analysis. Proc. of the ASME Turbo Expo 2011,
748 Vancouver (Canada), June 6-10; 2011. DOI: 10.1115/GT2011-45882
- 749 [50] Hoerner SF. Fluid-Dynamic Drag. Hoerner Fluid-Dynamics, 1965.
- 750 [51] Bianchini A, Ferrari L, Magnani S. On the effects of a skewed flow on the performance
751 of a three-bladed H-Darrieus turbine: experimental and theoretical analyses.
752 Proceedings of the International Conference on Applied Energy (ICAE) 2012, Suzhou
753 (China), July 5-8, 2012.