Available online at www.sciencedirect.com
SciVerse ScienceDirect
ELSEVIER
Procedia Computer Science 19 (2013) 728 – 735
Procedia
Computer Science

The 3rd International Conference on Sustainable Energy Information Technology (SEIT 2013)

Numerical investigation of a Darrieus rotor for low-head hydropower generation

Marco Torresi[a,*], Bernardo Fortunato[a], Sergio M. Camporeale[a]

[a]*Department of Mechanics, Mathematics and Management (DMMM), Politecnico di Bari, Via Re David, 200, Bari 70125, Italy*

Abstract

The aim of this paper is to numerically investigate the performance of a cross-flow water turbine of the Darrieus type for very low head hydropower applications. The interest for this kind of vertical axis turbine relies on its versatility. For instance, in the field of renewable energy, this kind of turbine may be considered for different applications, such as: tidal power, run-of-the-river hydroelectricity, wave energy conversion. Until now, low head hydropower, with heads less than 2 meters, has remained scarcely developed due to the relatively low energy density, which makes the cost of generation higher than traditional hydropower applications. However, in the spirit of distributed generation, the use of low head hydropower can be reconsidered, having the advantage of lower electricity transmission losses due to the localization near the consuming area. Nonetheless, it is fundamental to improve the turbine performance and to decrease the equipment costs for achievement of "environmental friendly" solutions and maximization of the "cost-advantage". In the present work, the commercial CFD code Fluent is used to perform 2D simulations, solving the incompressible Unsteady Reynolds-Averaged Navier-Stokes (U-RANS) equations discretized by means of a finite volume approach. The implicit segregated version of the solver is employed. The pressure-velocity coupling is achieved by means of the SIMPLE algorithm. The convective terms are discretized using a second order accurate upwind scheme, and pressure and viscous terms are discretized by a second-order-accurate centered scheme. A second order implicit time formulation is also used. Turbulence closure is provided by the realizable $k - \epsilon$ turbulence model. The model has been validated, comparing numerical results with available experimental data.

Keywords: Low-head hydropower, Darrieus-type rotor, CFD analysis, Turbine performance

1. Introduction

Fossil fuel depletion and greenhouse gas (GHG) emissions are sources of major concern all over the world. In order to mitigate these problems and to try to preserve the planet for the future generations, it is necessary to point on energy saving, efficiency and renewable power generation. Among renewables, a renewed interest is focused on hydropower generation (see [1], [2], [3]), which is characterized by a very low carbon footprint, mainly associated with the construction and decommissioning processes, whereas there are essentially no emissions of carbon dioxide

*Corresponding author. Tel.: + 39 080 5963577; fax: + 39 080 5963411
Email address: m.torresi@poliba.it (Marco Torresi)

doi:10.1016/j.procs.2013.06.096

(CO_2) during operation. Furthermore, whilst traditional hydropower based on storage schemes have a higher footprint ($10 - 30\, g\, CO_{2\,eq}/kWh$), due to the large amounts of the raw materials (e.g., steel and concrete) required to construct the dams, run-of-river schemes (typical for low-head hydropower), having very small reservoirs (those with weirs) or none at all, do not give rise to significant emissions during their operation (their carbon footprints are less than $5\, g\, CO_{2\,eq}/kWh$, some of the lowest of all electricity generation technologies [4]).

Nomenclature

B	$[m]$	duct and turbine heights
c	$[m]$	blade chord
$C_t = T/(1/2\,\rho\, B\, D\, V^{*2}\, R)$	[-]	torque coefficient
$D = 2R$	$[m]$	rotor diameter
H_t	$[m]$	head drop between up- and down-stream ponds
n	$[rpm]$	rotational speed
Q	$[m^3/s]$	flow rate
R	$[m]$	rotor radius
S	$[m]$	duct width
T	$[N\,m]$	shaft torque
T_b	$[N\,m]$	single blade torque contribution
$T^* = T/(1/2\,\rho\, U_{tip}^2\, R^3)$	$[-]$	non dimensional torque
$V_{in} = Q/(B\,S)$	$[m/s]$	inlet velocity
$V^* = Q/(B\,D)$	$[m/s]$	section-averaged velocity
$U_{tip} = \omega\, R$	$[m/s]$	tip speed or rotor peripheral velocity
$U^* = V_{in}/U_{tip}$	$[-]$	flow coefficient
$\Delta\, p^\circ$	$[Pa]$	total pressure drop across the rotor
$\Delta\, p^* = \Delta\, p^\circ/(1/2\,\rho\, U_{tip}^2)$	$[-]$	pressure drop coefficient
η	$[-]$	efficiency
θ	$[deg]$	azimuth angle
ρ	$[kg/m^3]$	air density
ω	$[rad/s]$	angular velocity

The worldwide technical potential for hydropower generation is estimated in $3721\, GW$ (which corresponds to $14576\, TWh/yr$) but only roughly a 25% is currently installed (actually the total installed hydropower capacity in 2009 was only $926\, GW$). However, the percentage of undeveloped capacity differs worldwide, ranging from about 47% in Europe and North America to 92% in Africa, which indicates large opportunities for continued hydropower development [5]. Moreover, a virtuous nation such as Canada, where hydropower generation is already well developed, nonetheless, is investing particularly in small hydropower (SHP). For instance, Natural Resources Canada is supporting the first North American demonstration of very low head hydropower at two sites in Canada [6]. Currently in the USA, hydropower accounts for about 6% of the electricity production. However, most of the larger and more traditional hydroelectric resources have already been developed [7]. Analogous situation can be found in the European Union (EU). Hence, in these nations, a clean energy rationale for development of small and low-head hydropower resources may now exist. For instance, a research undertaken under the European Union's Joule Programme has established that Europe's rivers represent a major renewable energy resource, with low-head hydropower capable of generating thousands of megawatts. Up to 70% of the resource would be cost-effective today with environmental electricity tariffs, already available in some EU countries, such as Germany, Denmark and the Netherlands. Although some small hydro manufacturers have developed successful innovations, the appropriate technology for low-head sites is still evolving. For this reason it is believed to be fundamental the intensification of R&D in the field of low-head hydropower generation. In order to develop effective methodologies to improve hydro turbine performance and to investigate innovative turbine concepts, computational fluid dynamics (CFD) can play a significant role. In this work, a ducted Darrieus-type hydro turbine has been considered for utilization of extra-low head hydropower (head less than $2\, m$). The turbine performance and the flow behavior have been evaluated by means of numerical simulations. The CFD model has been validated by comparing the numerical results with experimental data obtained by Shimokawa et al. [8] for the specific case of normal intake configuration. A plant schematic is reported in Fig. 1. The Darrieus rotor is a mechanical and structural simple machine. However, the mechanical simplicity doesn't extend to the rotor aerodynamics: the airfoils follow a circular path with a continuous cyclic variation in relative velocity and angle of attack. Hence, airfoils experience periodically variable loads, which are also affected by dynamic stall phenomena especially at low tip speed ratios. Moreover, the flow around the blades during the downstream passages can be disturbed by the wakes generated by the same blades during their upstream passages and by other elements (e.g., the turbine shaft). These features make the fluid dynamic analysis of such a turbine very challenging.

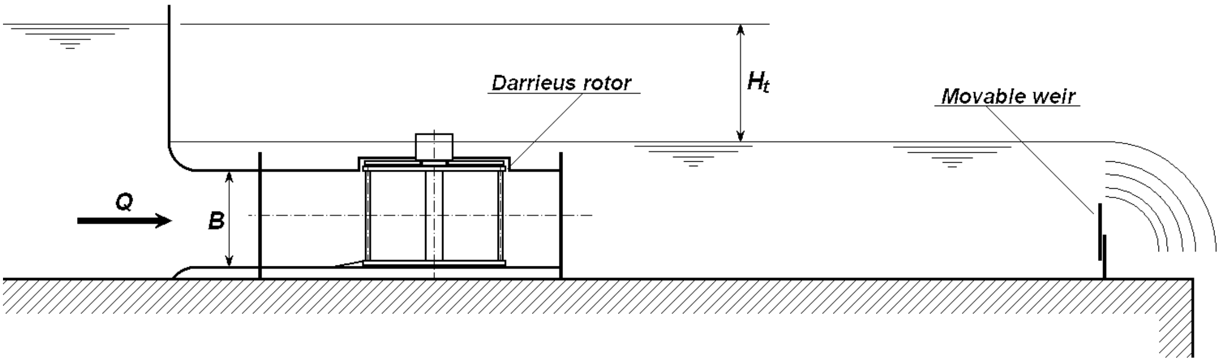

Figure 1: Plant schematic (based on [8])

2. Numerical model

In the numerical model end effects have been neglected performing only 2D simulations with the commercial CFD code Fluent [9]. This choice can be justified, considering that the turbine blades extend over the entire channel height o and are also equipped with end plates. However, the model is not actually able to take into account the development of the boundary layer on the upper and lower wall of the channel. The code was configured in order to solve the incompressible Unsteady Reynolds-Averaged Navier-Stokes (U-RANS) equations discretized by means of a finite volume approach. The implicit segregated version of the solver has been employed. The pressure-velocity coupling has been achieved by means of the SIMPLE algorithm. The convective terms have been discretized using a second order accurate upwind scheme, and pressure and viscous terms have been discretized by means of a second order accurate centred scheme. Furthermore, a second order implicit time formulation has been used. Time step has been defined in such a way that the turbine rotates by 1 *deg* each time performing 75 iterations per time step. Turbulence closure has been provided by the realizable $k-\epsilon$ turbulence model [10]. This means that the Reynolds's stresses are related to the mean velocity gradient according to the Boussinesq approach. Standard wall functions (based on the proposal of Launder and Spalding [11]) are solved near the duct walls and the blade surfaces. Actually the governing equations are solved in two different computational domains: a circular one, which includes the Darrieus rotor and rotates together with it; the other being stationary. Then, the sliding mesh model allows to compute the fluxes across the two non-conformal interfaces.

Table 1: Geometric characteristics of the system under investigation

Parameters	Symbol	Units	Value
Rotor Diameter	D	m	0.39
Chord to Radius ratio	c/R		0.30
Duct width	S	m	0.4112
Duct height	B	m	0.2

3. Computational domain

The numerical simulations have been performed on a straight bladed Darrieus rotor. The three blades of the turbine have profiles belonging to the symmetrical NACA four-digit series with a maximum thickness equal to 18% of the chord, i.e. a NACA 0018 profile. The main geometric characteristics are summarized in Table 1. Furthermore, the blades are tangential to the pitch circle at 50% chord points. Having decided to perform only 2D simulations, the computational domain has been simplified with respect to the actual test rig. A straight duct has been considered into which the Darrieus rotor can rotate (Fig. 2). No struts or shaft have been considered in the simulations. The grid has been developed through the use of the commercial software Gambit. The entire computational domain has been

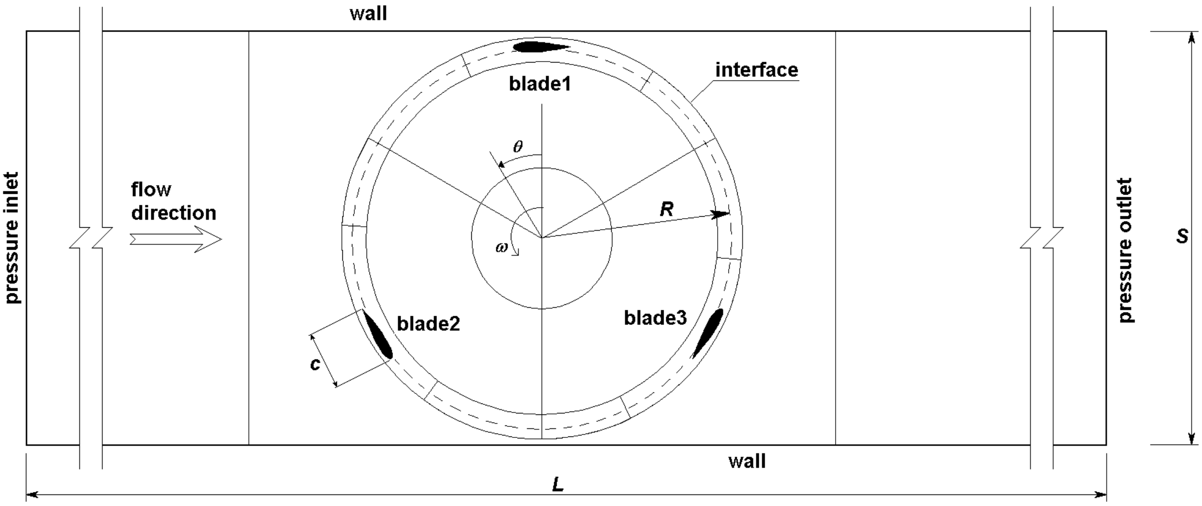

Figure 2: computational domain

divided into several parts due to the necessity of having zones with different mesh refinements (Fig. 3). In particular, a structured grid (not particularly dense) has been generated in the regions upstream and downstream of the rotor, which extend over 8 and 16 rotor diameters respectively, whereas an hybrid mesh has been generated in its vicinity. Around each blade, a fine structured grid (246 cells, 5 rows, first cell height equal to 0.23 mm, grow factor equal to 1.1) was used in order to correctly predict the flow behavior in the boundary layer region (Fig. 3). Similarly, a fine mesh on the duct walls has been generated in order to be able to better analyze the effect of the boundary layer in the region interposed between the rotating elements and the duct (Fig. 3). The structured grid upstream the rotor has 100 cells streamwise with the first cell near the rotor having a cell height equal to 5 mm. Downstram the only difference is that the number of cells is 140. Spanwise a symmetrical cell distribution is assigned with 64 cells with a first cell height near the duct walls equal to 1 mm. The mesh has globally 111000 cells but 86% of the cells is in the region close to the Darrieus rotor. Concerning the boundary conditions, at the duct inlet the assigned parameters are the total pressure (uniform along the boundary), and a uniform velocity distribution (normal to boundary), whereas at the duct outlet the assigned parameter is only the static pressure distribution (uniform along the boundary). At walls the no slip condition is satisfied. Blades have a null velocity relative to the circular moving mesh. Moving and stationary zones are separated by two coincident circular interfaces.

4. Results

Numerical simulations have been carried out imposing a constant rotational speed, $n = 245\,rpm$, in all the simulations. Hence, in order to guarantee the desired U_{tip}/V^* ratio, the head difference between the up- and down-stream ponds, H_t, has been opportunely assigned. Four simulations have been performed considering the following flow coefficients: 1) $U^* = 0.165$, 2) $U^* = 0.200$, 3) $U^* = 0.248$, 4) $U^* = 0.293$, or in terms of U_{tip}/V^* ratios: 1) 5.6, 2) 4.6, 3) 3.7, 4) 3.2. The solutions have been initialized by imposing uniform flow conditions computed from the inlet

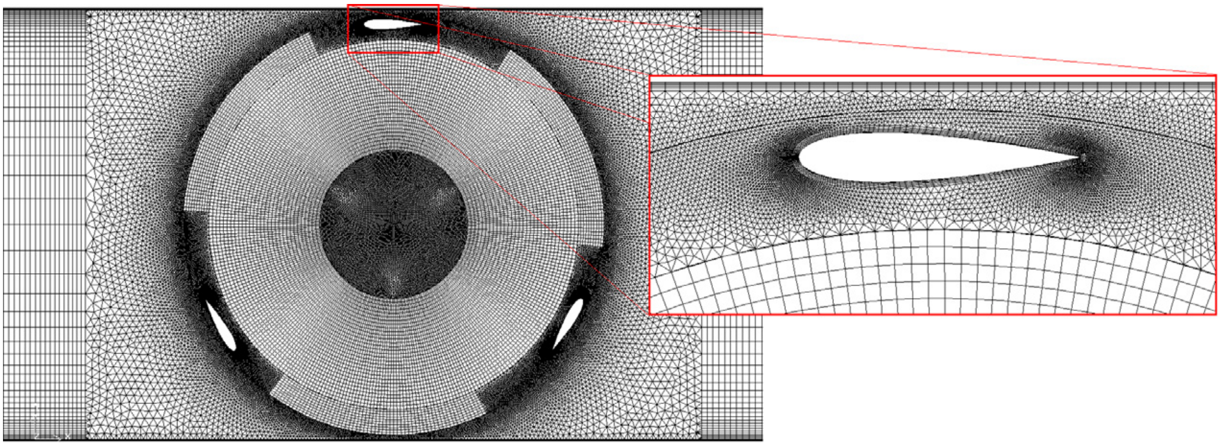

Figure 3: Details of the mesh in the zone swept by the Darrieus rotor

boundary. In order to reach solution convergence, 50 rotor revolutions have been considered. This high number of revolutions has been necessary in order to let the turbine wake reach the domain outlet, which occurs only after the first 30 revolutions, and then to definitely reach periodicity (as shown in Fig. 4.a for the case with $U_{tip}/V^* = 3.7$). In Fig. 4.b the torque coefficients of each blade and of the entire rotor, computed during the last 2 revolutions, are reported confirming that the solution has become periodic. Furthermore, in Fig. 4.c the torque coefficients of the three blades are reported vs. the azimuth angle. The three curves are undistinguishable, which means that all the three blades behave in the same way and hence the rotor has not only a geometric but also a fluid dynamic periodicity of $120\,deg$.

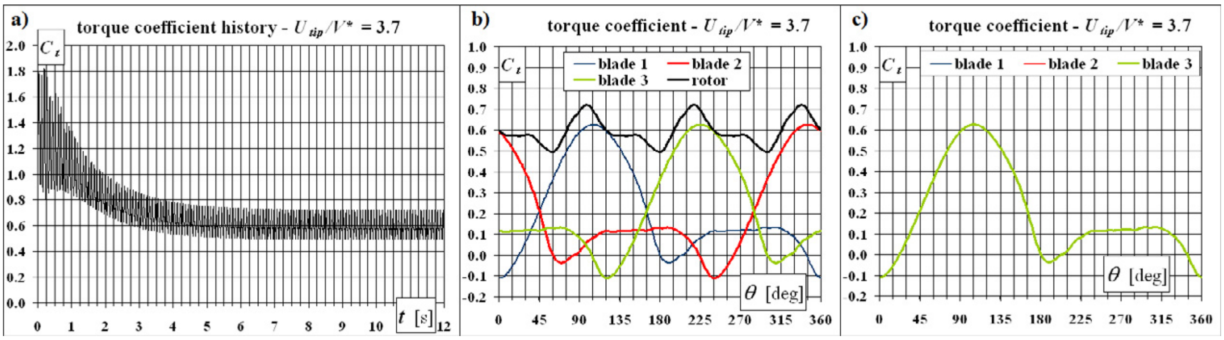

Figure 4: a) Shaft Torque coefficient history (first 50 revolutions); b) torque coefficients during one revolution; c) superposition of the torque coefficients of the three blades at the same azimuth position.

In Fig. 5, the rotor performance are reported in terms of pressure drop coefficient, Δp^*, non dimensional torque, T^*, and efficiency, η vs. flow coefficient, U^*. These values have been averaged over a single period. The numerical results and the experimental data are in very good agreement, especially in terms of non dimensional torque and pressure drop coefficient, whereas the efficiency evidences small discrepancies. Nevertheless, considering the approximations introduced by performing 2D simulations (hence neglecting end effects) and neglecting some rotor details (such as the rotor shaft and the blades struts), the validity of the proposed numerical model (dealing with the performance analyses of a cross flow turbine confined inside a duct) can be substantially confirmed. The pressure drop coefficient linearly varies with respect to the flow coefficient, whereas the non dimensional torque (in the range considered) increases more than linearly with respect to the flow coefficient. In order to explain this behavior, the operating principles of a Darrieus must be considered: when the flow coefficient is low, the blade relative velocity is dominated by the peripheral velocity, which implies that the angle of attack, even if variable, is generally low and hence the lift force is mainly directed radially whilst the drag force is mainly tangential, reducing torque. When the flow coefficient increases, performance drops due to significant flow separation on the blades but this flow configurations have not been taken into account in this work.

When considering local blade performance (Fig. 6), the different behavior of the blades from upwind to downwind passages is evident. In fact, during the upwind passage the blade gives the best performance. Starting from the zero azimuth angle, when the blade is perfectly aligned with the main flow and the torque coefficient is consequently negative, the blade experiences a continuous increase in terms of torque coefficient until a maximum is reached when the azimuth angle is around 90 *deg* (this happens since no significant blade separation occurs, being small the angle of attack). Since the blades perform well during the upwind passage, they extract a significant amount of energy from the flow, hence the blade performance decays during the downwind passage. This means that, in order to improve the system performance, attention must be focused particularly on the blade upwind passage. For instance, Shimokawa et al. [8] proposed to introduce a narrow inlet nozzle just upstream the rotor in order to increase the flow velocity (hence concentrating the flow energy) where the blades perform at their best. Another possible modification could be the changing of the blade stagger angle or the use of non symmetric blade profiles in order to improve the blade performance during the upstream passage but limiting the worsening during the downstream passage.

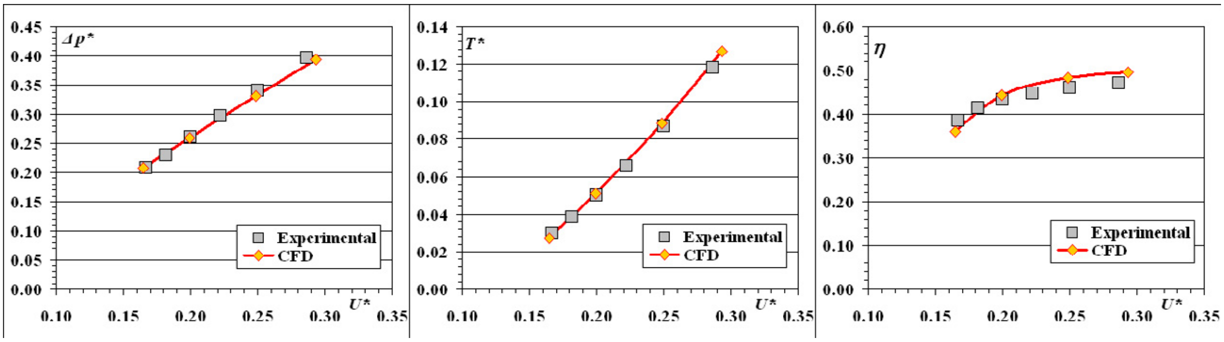

Figure 5: Turbine performance in terms of non-dimensional parameters (from left to right: Δp^*, T^*, η vs. U^*)

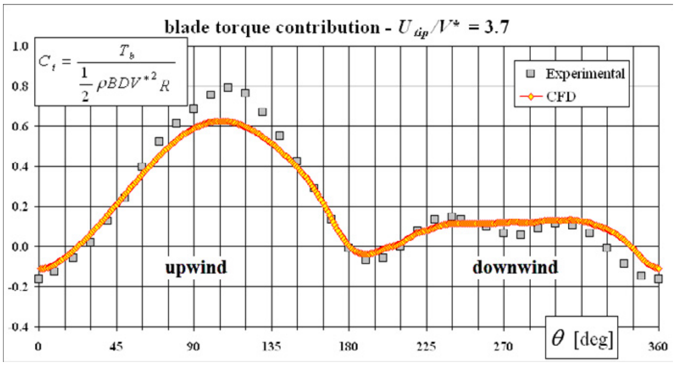

Figure 6: Blade torque coefficient, C_t, vs. azimuth angle, θ, at $U_{tip}/V^* = 3.7$

In order to analyse the flow field, the contours of the vorticity magnitude (Fig. 7.a) and velocity magnitude (Fig. 7.b) are considered at two different U_{tip}/V^* ratios (3.7 and 5.7). Particularly effective are the vorticity magnitude contours in order to evidence the strong interaction between the blades and the duct walls. Actually, at each passage, the blades disturb the boundary layers near the duct walls. Another specific flow feature is the formation and shedding of separation bubbles and their corresponding vortices, which are generated when the blades, starting their downwind passages ($\theta > 180\,deg$), interact with the preceding blade wakes. It is evident that, when the flow coefficient, $U*$, is higher and hence the U_{tip}/V^* ratio is lower ($U_{tip}/V^* = 3.7 < U_{tip}/V^* = 5.6$) the vortex are shed

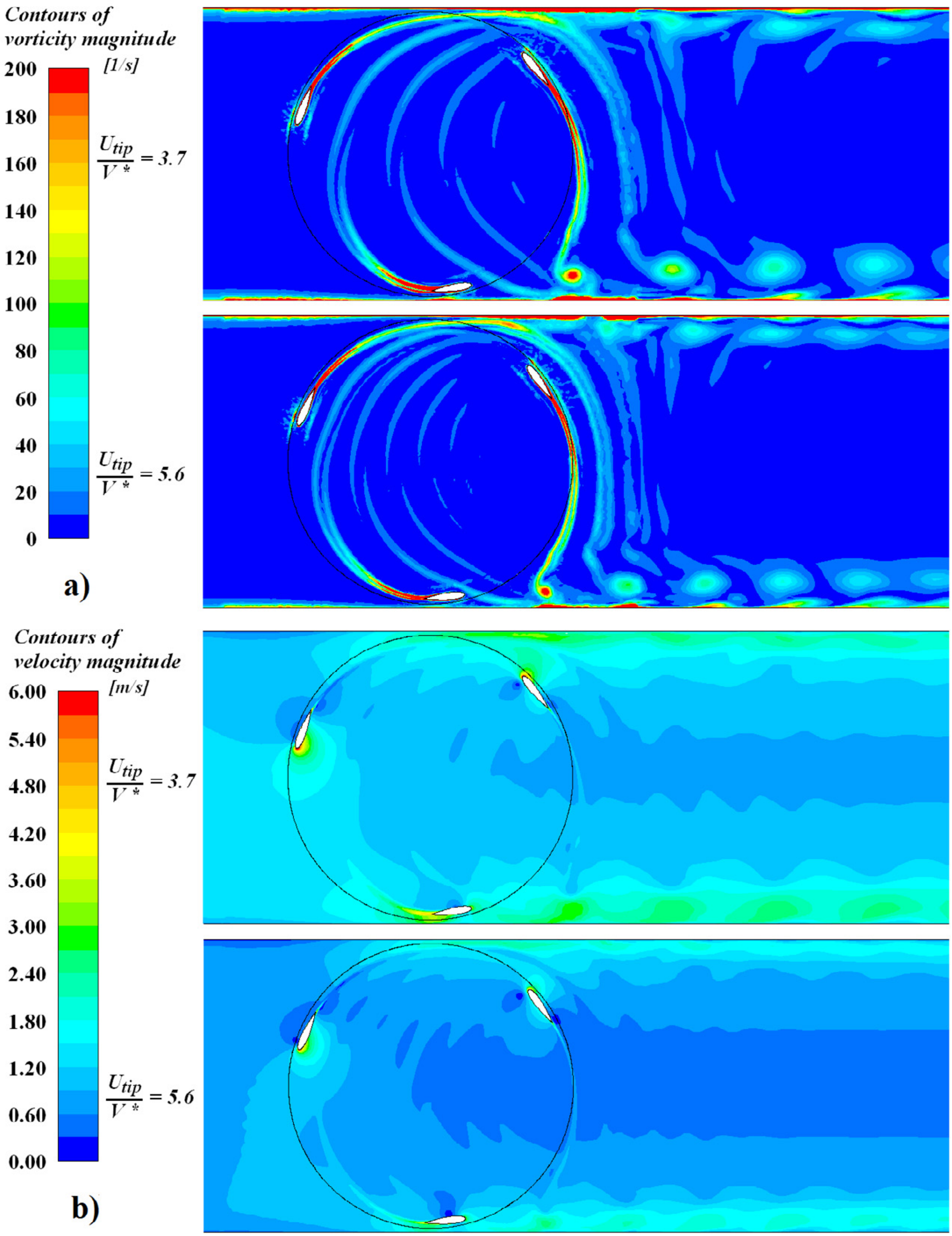

Figure 7: Contours of vorticity (a) and velocity (b) magnitude at two different U_{tip}/V^* (top: $U_{tip}/V^* = 3.7$, bottom: $U_{tip}/V^* = 5.6$)

at higher speed increasing their distances. Velocity magnitude contours evidence the flow acceleration in the small gaps between the rotor and the duct walls. This phenomenon is more intense at low U_{tip}/V^* ratios when the rotor determines an increase of the pressure drop across the duct and hence for the flow it is easier to bypass the rotor rather the passing through it.

5. Conclusions

The performance of a cross-flow water turbines of the Darrieus type, for very low head hydropower applications, has been numerically investigated after having validated the CFD model by comparison of the results with experimental data from Shimokawa et al. [8]. The numerical results are in very good agreement in terms of global parameters, above all non dimensional torque and pressure drop coefficient but also in terms of local performance such as blade torque contribution at different azimuth angles. From the analysis of the blade torque contribution during the rotor revolution, the blade upwind passage is the part of the blade operation where the attention must be focused in order to improve global rotor performance. In this preliminary work, some fluid dynamic characteristics (e.g. end effects) and some rotor details (such as the rotor shaft and the blades struts) have been neglected, which have caused some discrepancies in the evaluation of the turbine efficiency.

Aknowledgements

We would like to express thanks to Michele Lacerenza and Angelo Colucci, for their invaluable help.

References

[1] L. Kosnik, The potential for small scale hydropower development in the us, Energy Policy 38 (10) (2010) 5512 – 5519. doi:10.1016/j.enpol.2010.04.049.

[2] S. Singal, R. Saini, C. Raghuvanshi, Analysis for cost estimation of low head run-of-river small hydropower schemes, Energy for Sustainable Development 14 (2) (2010) 117 – 126. doi:10.1016/j.esd.2010.04.001.

[3] A. T. de Almeida, C. Inverno, J. L. de Almeida, J. A. S. Marques, B. Santos, Small-hydropower integration in a multi-purpose dam-bridge for sustainable urban mobility, Renewable and Sustainable Energy Reviews 15 (9) (2011) 5092 – 5103. doi:10.1016/j.rser.2011.07.047.

[4] U. P. O. of Science, Technology, ”carbon footprint of electricity generation”, science and technology postnote # 268 (2006). URL `http://www.parliment.uk/documents/upload/postpn268.pdf`

[5] A. Kumar, T. Schei, A. Ahenkorah, R. C. Rodriguez, J.-M. Devernay, M. Freitas, D. Hall, . Killingtveit, Z. Liu, Special Report of the Intergovernmental Panel on Climate Change, Cambridge University Press, New York, 2012, Ch. Hydropower.

[6] S. Cooke, C. D. of Fisheries, Oceans, Literature Review, Synthesis and Proposed Guidelines Related to the Biological Evaluation of ”fish Friendly” Very Low Head Turbine Technology in Canada, Canadian technical report of fisheries and aquatic sciences, Department of Fisheries and Oceans, 2011.

[7] R. Campbell, Small hydro and low-head hydro power technologies and prospects, Congressional Research Service 7-5700, R41089.

[8] K. Shimokawa, A. Furukawa, K. Okuma, D. Matsushita, S. Watanabe, Experimental study on simplification of darrieus-type hydro turbine with inlet nozzle for extra-low head hydropower utilization, Renewable Energy 41 (0) (2012) 376 – 382. doi:10.1016/j.renene.2011.09.017.

[9] FLUENT: Users Guide 6.3, Fluent Inc., Centerra Resource Park 10, Cavendish Court, Lebanon, NH 03766, USA, 2006.

[10] T. Shih, W. Liou, A. Shabbir, Z. Yang, J. Zhu, New k-e eddy-viscosity model for high reynolds number turbulent flows - model development and validation, Computers Fluids 24 (3) (1995) 227–238.

[11] B. Launder, D. Spalding, The numerical computation of turbulent flows, Computer Methods in Applied Mechanics and Engineering 3 (1974) 269–289.