Effect of the shaft on the aerodynamic performance of urban vertical axis wind turbines

Citation for published version (APA):

Document license:
CC BY

DOI:
10.1016/j.enconman.2017.07.055

Document status and date:
Published: 05/08/2017

Document Version:
Publisher’s PDF, also known as Version of Record (includes final page, issue and volume numbers)

Please check the document version of this publication:

• A submitted manuscript is the version of the article upon submission and before peer-review. There can be important differences between the submitted version and the official published version of record. People interested in the research are advised to contact the author for the final version of the publication, or visit the DOI to the publisher’s website.
• The final author version and the galley proof are versions of the publication after peer review.
• The final published version features the final layout of the paper including the volume, issue and page numbers.

Link to publication

General rights
Copyright and moral rights for the publications made accessible in the public portal are retained by the authors and/or other copyright owners and it is a condition of accessing publications that users recognise and abide by the legal requirements associated with these rights.

• Users may download and print one copy of any publication from the public portal for the purpose of private study or research.
• You may not further distribute the material or use it for any profit-making activity or commercial gain
• You may freely distribute the URL identifying the publication in the public portal.

If the publication is distributed under the terms of Article 25fa of the Dutch Copyright Act, indicated by the “Taverne” license above, please follow below link for the End User Agreement:
www.tue.nl/taverne

Take down policy
If you believe that this document breaches copyright please contact us at:
openaccess@tue.nl
providing details and we will investigate your claim.

Download date: 22. Apr. 2019
Effect of the shaft on the aerodynamic performance of urban vertical axis wind turbines

Abdolrahim Rezaeiha, Ivo Kalkman, Hamid Montazeri, Bert Blocken

ARTICLE INFO
Keywords: Vertical axis wind turbine (VAWT) Shaft (tower) Performance improvement CFD URANS Rotating rough cylinder

ABSTRACT
The central shaft is an inseparable part of a vertical axis wind turbine (VAWT). For small turbines such as those typically used in urban environments, the shaft could operate in the subcritical regime, resulting in large drag and considerable aerodynamic power loss. The current study aims to (i) quantify the turbine power loss due to the presence of the shaft for different shaft-to-turbine diameter ratios $\delta$ from 0 to 16%, (ii) investigate the impact of different operational and geometrical parameters on the quantified power loss and (iii) evaluate the impact of the addition of surface roughness on turbine performance improvement. Unsteady Reynolds-averaged Navier-Stokes (URANS) calculations are performed on a high-resolution computational grid. The evaluation is based on validation with wind-tunnel measurements. The results show that the power loss increases asymptotically with increasing $\delta$ due to the higher width and length of the shaft wake as the blades pass through a larger region with lower velocity in the downwind area. A maximum power loss of 5.5% compared to the hypothetical case without shaft is observed for $\delta = 16\%$. The addition of surface roughness is shown to be an effective approach to shift the flow over the shaft into the critical regime, reducing the shaft drag and wake width as a result of a delay in separation. For an optimal dimensionless equivalent sand-grain roughness height of 0.08, the turbine power coefficient at $\delta = 4\%$ improves by 1.7%, which is equivalent to a 69% recovery of the corresponding turbine power loss. The results are found to be virtually independent of the shaft-to-turbine rotational speed ratio.

1. Introduction

Vertical axis wind turbines (VAWTs) have regained interest during the last decade for application as large-scale multi-MW turbines in offshore areas [1–3] and as small-scale turbines in urban environments [4,5]. The large-scale turbines are interesting for off-shore application due to their low manufacturing, installation and maintenance costs, scalability, robustness, reliability, and installation of the generator on the ground, while their omni-directional capabilities make them highly desirable for urban environments where the wind direction is frequently changing [6]. However, a significantly lower amount of research in the past three decades has resulted in VAWT performance falling behind that of their horizontal axis counterparts. Several research activities have recently focused on further understanding the complex unsteady aerodynamics of VAWTs [7–15] and characterizing their performance via parameters such as the number of blades [16], blade (airfoil) shape [17–20], turbine solidity [16,18,21] and blade pitch angle [22,23]. Employment of ducts [11], guide vanes [24–26] and utilization of flow control on turbine blades [27–31] also have recently received attention.

The tower is an inseparable component of horizontal and vertical axis wind turbines, which greatly affects the flow on the blades in its neighborhood. For an upwind horizontal axis wind turbine (HAWT) the effect is due to the blades passing through the stagnation region in front of the tower, while for a VAWT or a downwind HAWT this is due to the blades passing through the wake of the tower. A significant amount of research has been performed to characterize the effect of the tower shadow on the aerodynamic and structural performance of HAWTs [32–36]. In contrast, early research on VAWT focused on large turbines with a very small ratio of the tower diameter to turbine diameter. This results in a large relative distance between the tower (which can also be referred to as the shaft) and the blades, in which the effect of the shaft wake on the turbine aerodynamic performance was assumed to be negligible and was therefore not quantified in existing research [4]. However, this is not the case for smaller urban-scale VAWTs with comparatively large shaft-to-turbine diameter ratios, as the turbine power loss due to the blades passing through the wake of the shaft could be substantial. However, to the best of our knowledge, a detailed
The current study is performed in three steps: (1) The turbine power loss associated with the turbine shaft for different shaft-to-turbine diameter ratios $\delta$ from 2% to 16%, relevant for small- to medium-scale VAWTs, is quantified. The results are then compared with those for a hypothetical turbine with no shaft. (2) The impact of different operating conditions (including inlet turbulence intensity, chord-based Reynolds number and tip speed ratio) and geometrical characteristics (including solidity and number of blades) on the quantified power loss is investigated. (3) The flow regime over the turbine shaft is identified. Then the effect of the shaft surface roughness is investigated in order to find the optimal roughness height corresponding to the critical Reynolds number $Re_{crit}$, where a minimum shaft drag is achieved. The corresponding effect on the turbine aerodynamic performance is then investigated in detail. The dependence of the performance of the turbine with optimal roughness height on the shaft-to-turbine rotational speed ratio $\eta$ is then studied in the range from 0 to 1, where 0 corresponds to a stationary shaft and 1 to a shaft rotating at the same rotational speed and direction as the turbine.

Unsteady Reynolds-averaged Navier-Stokes (URANS) simulations are performed on a high-resolution computational grid. The evaluation is based on validation with wind-tunnel measurements of flow over smooth and rough cylinders by [40,41,48], and for a VAWT by [49,50].

The outline of the paper is as follows: Section 2 presents a description of the computational settings and parameters where the geometrical and operational characteristics, computational domain, grid and other numerical settings and validation studies for both (smooth and rough) cylinders and the turbine are presented. The results of the study on the quantification of the turbine power loss due to the presence of the shaft for different shaft-to-turbine diameter ratios $\delta$ are discussed in Section 3.1. Section 3.2 presents a sensitivity analysis to investigate the effect of different operational and geometrical parameters on the turbine power loss due to the presence of the shaft. The possibility of turbine power improvement by addition of surface roughness to the shaft is discussed in Section 4.1. Section 4.2 explains the effect of the shaft-to-turbine rotational speed ratio $\eta$ on the findings. The limitations of the work, recommendations for future work and the conclusions are provided in Section 5.

2. Computational settings and parameters

Given the main objectives of this study, it is important first to make sure that the flow around the shaft of the turbine, which is basically a cylinder (either smooth or rough), as well as the flow around the whole turbine is accurately simulated. Therefore, three sets of validation studies are performed where the geometrical and operational characteristics are selected to be the same or similar to those in the main study. The studied cases are (1) a stationary smooth cylinder at subcritical $Re_D (= DU_{\infty}/\nu)$; (2) a stationary rough cylinder at subcritical and critical $Re_D$ and (3) a VAWT. CFD results of the VAWT have been extensively validated against experimental data of Tescione et al. [50] and the results, together with a comprehensive discussion of the possible explanations for observed deviations, have been published in [51].
A second validation study for a VAWT using similar computational settings has been performed and published in [23] where turbine $C_p$ values at different tip speed ratios are compared against experimental data by Castelli et al. [49]. For brevity, these results and discussions are not repeated here. Instead, the simulations reported in Section 2.2.4 of this paper are solely intended to show that the turbulence model employed in the present study for the prediction of flow over the VAWT, namely the 3-equation $k$-$\omega$ SST model with intermittency transition submodel ($k$-$\omega$ SST int.), performs comparably to the already validated 4-equation transition SST model at the same operating conditions while reducing the calculation cost by solving one less equation. The geometrical and operational characteristics for each set of simulations are explained in this section.

2.1. Cylinder

2.1.1. Geometrical and operational characteristics

Two sets of validation studies are performed to provide confidence in the adopted approach for accurate simulation of the flow around both smooth and rough cylinders. The main geometrical and operational characteristics of the two studies are described in Table 1. As it will be discussed in Section 3, the smooth shaft of an urban-scale VAWT operates in the subcritical regime. Therefore, the validation study on a smooth cylinder also focuses on this regime. Note that the addition of surface roughness to the shaft can shift the flow over the turbine shaft from the subcritical to the critical regime. In the validation study for the rough cylinder, therefore, different roughness heights with $Re_D$ corresponding to the subcritical and critical regimes are employed. As $Re_{crit}$ decreases with increasing roughness height (see Eq. (1) [52,53]), lower $Re_D$ is selected for the highest roughness height to keep the flow regime critical. The critical regime is chosen to maximize the similarity between the validation study on the rough cylinder and the flow regime over the rough turbine shaft, which is subcritical to critical.

$$Re_{crit} = \frac{6000}{\sqrt{k_f/D}}$$

(1)

2.1.2. Computational domain and grid

The two-dimensional computational domain employed for the simulation of flow around the cylinder (Fig.1a) has the origin at the cylinder center. The distance from the cylinder center to the domain inlet and outlet is $10D$ and $15D$, respectively, and the domain width is $20D$. A fully structured grid (Fig. 1b and c) is generated where the maximum $y^+$ over the cylinder is less than 4 and the total number of cells is approximately 120,000. The boundary layer grid over the cylinder is shown in Fig.1d.

2.1.3. Other numerical settings

In the simulations, the mean velocity and turbulent intensity inlet boundary conditions are uniform profiles according to the measured data. The approach-flow total turbulence intensity at the domain inlet is 0.6% and 0.7% for the smooth and rough cylinders while the incident total turbulence intensity is 0.57% and 0.67%, respectively [51,54,55]. Zero static gauge pressure is applied at the outlet plane. A symmetry condition on the sides and a no-slip condition on the cylinder wall is employed.

URANS calculations are performed with second-order spatial and temporal discretization and SIMPLE pressure-velocity coupling using the commercial flow solver ANSYS/Fluent 16.1 [56]. The calculations are initialized with steady-state RANS. The time step for the unsteady calculations is 0.001 s with 20 iterations per time step. Data sampling is initiated after 6000 time steps (6 s), which is equal to more than 10 flow-through times. The initialization time is found to be sufficient for the cylinder drag coefficient to reach a statistically steady state for all the tested cases. The sampled data are then averaged over 4000 time steps (4 s). Note that the averaging period is selected to correspond to more than 15 vortex shedding cycles for all the cases [57]. The modeling of turbulence is done using a 3-equation turbulence model, i.e. $k$-$\omega$ SST with the intermittency transition model ($k$-$\omega$ SST int.) [58]. This model solves an additional equation for intermittency $\gamma$ over those of the popular 2-equation $k$-$\omega$ SST turbulence model to provide a better prediction of the laminar-to-turbulent transition. Intermittency-based turbulence models have been shown to perform well for wall-bounded flows where the development of the boundary layer and the laminar-to-turbulent transition play an important role in the overall flow characteristics, e.g. flow over airfoils and cylinders [59,60].

In ANSYS/Fluent, for $\omega$-based turbulence models, the surface roughness is modeled using the modified law-of-the-wall for roughness developed by Cebeci and Bradshaw [61] based on Nikuradse’s data on flow in sand-grain roughened pipes [62]. For these turbulence models, a singularity in the modified law-of-the-wall can occur for large roughness heights at low values of $y^+$. In order to avoid this issue, the wall is virtually shifted by half of the height of the roughness elements, following the work by Schlichting [63], which proposes that for tightly-packed uniform sand-grain roughness the blockage effect induced in the boundary layer is equivalent to 50% of the height of the roughness elements. Therefore, this virtual shift in the wall distance by half the roughness height results in a corrected value of $y^+$ for the first cell adjacent to the wall, thus avoiding the singularity. A roughness constant $C_r$ of 0.5 is employed, representing a tightly-packed uniform sand-grain roughness [56]. Further details on the modeling of roughness can be found in Schlichting [63].

2.1.4. Validation study: Comparison of CFD results and wind-tunnel experiments

The results of the present CFD simulations of flow over a smooth cylinder in the subcritical regime ($Re_D = 1.4 \times 10^5$) are compared against the experiment by Cantwell and Coles [48] and the URANS (with the $k$-$\varepsilon$ RNG turbulence model) and LES calculations by Tutar and Holdø [64]. The results are provided in Table 2. The comparison shows a very good agreement between the present CFD results and the experimental data, within 3% for both drag coefficient and Strouhal number. As shown by Palkin et al. [57], the prediction of the laminar-to-turbulent transition is very important to predict the flow on cylinders accurately. The intermittency-based turbulence models can, therefore, be very suitable in this respect. The current study provides a better prediction of the drag coefficient than the LES study by Tutar and Holdø [64], whereas the LES study had a slightly lower deviation from the experiment for the Strouhal number. The former can be related to the high sensitivity of LES to grid quality, which might result in URANS outperforming LES on coarser grids [65-67].

Fig. 2 presents the profiles of the time-averaged normalized streamwise velocity along the symmetry line $y/D = 0$, in the wake of the smooth cylinder. A very good agreement between the present CFD study and the experiment in the prediction of the wake is observed. In this case, the average absolute deviation is about 0.04.

Results of the simulation of flow over a rough cylinder with different roughness heights $(1.1 \times 10^{-3} - \leq k_f/D \leq 3.0 \times 10^{-3})$ in the subcritical and critical regime are compared against the experimental data of Achenbach [40] and Achenbach and Heinecke [41] in Table 3. A
good agreement is observed for both drag coefficient and Strouhal number, with a maximum deviation of 9.4% for the drag coefficient and 8.3% for the Strouhal number. Moreover, the trend of drag coefficient as a function of \( k_s/D \) is quite well produced. In this case, the drag coefficient reduces as \( k_s/D \) increases from \( 1.1 \times 10^{-3} \) to \( 9.0 \times 10^{-3} \), and then increases with further increase in \( k_s/D \) (from \( 9.0 \times 10^{-3} \) to \( 30.0 \times 10^{-3} \)).

2.2. VAWT

2.2.1. Geometrical and operational characteristics

Flow around a two-bladed H-type VAWT with a solidity of 0.12, a diameter of 1 m, a height of 1 m and a swept area of 1 m\(^2\) is simulated. The turbine blade section is a NACA0018 airfoil with a chord of 0.06 m. The turbine has a smooth shaft with a diameter of 0.04 m, which is rotating in the same direction as the turbine with the same rotational speed, resulting in a shaft-to-turbine diameter ratio \( \delta \) of 4% and a shaft-to-turbine rotational speed \( \eta \) of 1, where \( \delta \) and \( \eta \) are given by Eqs. (2) and (3), respectively. The turbine operates at a moderate tip speed ratio of 4.5. The freestream velocity is 9.3 m/s with 5% turbulence intensity, and the turbine rotational speed is 84 rad/s (800 rpm). The geometrical and operational characteristics of the turbine are described in Table 4 where the airfoil chord-based Reynolds number \( R_e_c \) is defined using Eq. (4). It is important to note that \( R_e_c \) is almost an order of magnitude lower than \( R_e \). In practice the experienced streamwise velocity upstream of the shaft is lower than the free stream velocity due to the energy extraction by blades passing on the turbine fore half (\( 0^\circ \leq \theta \leq 180^\circ \)), leading to even lower \( R_e_c \) values.

\[
\delta = \frac{d_s}{d}
\]

(2)

\[
\eta = \frac{\Omega_s}{\Omega}
\]

(3)

\[
R_{e_c} = \left( \frac{c \sqrt{(R \Omega)^2 + U_m^2}}{y} \right)
\]

(4)

2.2.2. Computational domain and grid

The simulations are performed in a two-dimensional domain. Note that an earlier study by Rezaeia et al. [51] shows that using two- and three-dimensional domains leads to a negligible difference (< 1%) in the calculated turbine performance for the studied operating conditions. The main reason for this is that the low solidity and moderate tip speed ratio of the turbine minimize the complex 3D flow features associated with dynamic stall and blade-wake interactions. The computational domain consists of a rotating core (to facilitate the rotation of the cylinder) and a stationary domain.

---

**Table 2**

Comparison of present CFD results and experimental results [48] of drag coefficient and Strouhal number for a smooth cylinder in the subcritical regime (\( R_e_D = 1.4 \times 10^5 \)).

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Exp. [48]</th>
<th>( k-\omega ) SST int.</th>
<th>LES [64]</th>
<th>( k-\varepsilon ) RNG [64]</th>
</tr>
</thead>
<tbody>
<tr>
<td>( C_d )</td>
<td>1.23</td>
<td>1.19</td>
<td>−3.2</td>
<td>1.40</td>
</tr>
<tr>
<td>( St )</td>
<td>0.179</td>
<td>0.185</td>
<td>3.3</td>
<td>0.184</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td>0.167</td>
</tr>
</tbody>
</table>

---

Fig. 1. (a) Schematic of the computational domain; computational grid (b) in the whole domain, (c) near the cylinder and (d) the boundary layer grid for simulation of flow over a smooth cylinder. A similar domain and grid are used for the rough cylinder.

Fig. 2. Time-averaged normalized streamwise velocity along symmetry line \( y/D = 0 \) in the wake of a smooth cylinder in the subcritical regime (\( R_e_D = 1.4 \times 10^5 \)).
showed a negligible coarsening and re-study using 3 grids generated based on an overall linear factor of 400,000 cells. The grid resolution resulted from a grid-convergence employed on the walls. The two-dimensional grid has approximately 20 prove-sides and a no-slip condition on the airfoil and shaft walls. The approach-flow uniform total turbulence intensity at the domain inlet is 3.5 × 10⁻³, which is connected to the surrounding turbulence. The grid resolution resulted from a grid-convergence study can be found in Ref. [51].

Table 3

<table>
<thead>
<tr>
<th>( \frac{k}{D} )</th>
<th>( R_{o} )</th>
<th>( R_{e} )</th>
<th>Parameter</th>
<th>( \exp{\alpha} )</th>
<th>( k-\omega ) SST int.</th>
</tr>
</thead>
<tbody>
<tr>
<td>( 1.1 \times 10^{-3} )</td>
<td>79,000</td>
<td>200,000</td>
<td>( C_{D} )</td>
<td>1.04</td>
<td>0.99</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>( \sigma )</td>
<td>2</td>
<td>0.201</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>( St )</td>
<td>0.85</td>
<td>0.93</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>( St )</td>
<td>0.24</td>
<td>0.22</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>( C_{D} )</td>
<td>0.80</td>
<td>0.84</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>( St )</td>
<td>0.243</td>
<td>0.246</td>
</tr>
</tbody>
</table>

Comparison of present CFD results and experimental results [40,41] of drag coefficients of the turbine, which is connected to the surrounding fixed domain using a non-conformal interface with sliding grid (Fig. 3). The distances from the turbine center to the domain inlet and outlet are 10d and 25d, respectively, with a domain width of 20d (blockage ratio 5%). The diameter of the rotating core is 1.5d. The domain size is selected according to the guidelines for VAWTs identified in an extensive systematic sensitivity study [51].

The computational grid consists of quadrilateral cells, as shown in Fig. 4, where a boundary layer grid with a maximum \( y^+ \) below 4 is employed on the walls. The two-dimensional grid has approximately 400,000 cells. The grid resolution resulted from a grid-convergence study using 3 grids generated based on an overall linear factor \( \sqrt{2} \) for coarsening and refining [51]. The results of the grid-convergence study showed a negligible difference (< 1%) in the turbine power, and thrust coefficients \( (C_{P} \) and \( C_{T} \)) obtained on the three grids. The Grid Convergence Index (GCI) [68] was employed on \( C_{P} \) to quantify the discretization error for the coarse-medium grid pair using a safety factor \( (F_{s}) \) of 1.25, resulting in \( GCI_{fine} = 6.1 \times 10^{-3} \) (< 1.5% change in \( C_{P} \)) and \( GCI_{coarse} = 3.5 \times 10^{-3} \) (< 1% change in \( C_{P} \)). Therefore, the coarse grid selection is present for the selected study. Further details of the grid-convergence study can be found in Ref. [51].

2.2.3. Other numerical settings

The boundary conditions are a uniform constant velocity of 9.3 m/s at the inlet, zero static gauge pressure at the outlet, symmetry on the sides and a no-slip condition on the airfoil and shaft walls. The approach-flow uniform total turbulence intensity at the domain inlet is 5%, while the incident turbulence intensity is 4.42% [51,54,55].

URANS calculations are performed using the commercial CFD software package ANSYS/Fluent 16.1 [56]. Both spatial and temporal discretization is second-order, and pressure-velocity coupling is performed with the SIMPLE scheme. The solution from the steady RANS simulation is used for initialization after which 20 revolutions of the turbine are needed to reach a statistically steady state. Results are averaged over 10 revolutions where stated. The azimuthal increment (\( \delta \)) utilized for the unsteady simulations is 0.1° with 20 iterations per time step to ensure all scaled residuals drop below \( 10^{-5} \). The number of turbine revolutions in the initialization and data sampling phases and the employed azimuthal increment are selected according to the minimum requirements identified by Rezaeiha et al. [51,69]. Turbulence is modeled using two intermittency-based models: the 3-equation \( k-\omega \) SST model with the intermittency transition submodel (\( k-\omega \) SST int.) and, for the validation study only, the 4-equation transition SST turbulence model [58]. Both models are designed to provide a better prediction of the laminar-to-turbulent transition onset for wall-bounded flows as discussed in Section 2.1.3 [59]. The 4-equation turbulence model is already extensively validated [23,51] and the 3-equation model is shown (see Section 2.2.4) to have comparable performance with lower computational cost due to solving one less equation. The modeling of surface roughness for the turbine is the same as explained in Section 2.1.3 for the cylinder.

2.2.4. Validation study: Comparison of CFD results and wind-tunnel experiments

Table 5 presents the time-averaged (over 10 turbine revolutions) normalized streamwise and lateral velocity components at different downflow locations of the turbine (\( 2 \leq x/R \leq 4 \); see Fig. 5). The results obtained by the 3-equation \( k-\omega \) SST model with the intermittency transition submodel shows a lower average deviation from the experimental data than those by the transition SST model. The maximum average deviations for the streamwise velocity, which occurs at \( x/R = 4 \), are 12.6% and 16.4%, respectively. The turbine shaft \( C_{D} \) calculated using the 4-equation and 3-equation models in the two-di-mensional simulations are 1.24 and 1.22, respectively. Zheng and Lei [60] showed that coupling of the intermittency transition model with the \( k-\omega \) SST model improves the prediction of the flow pattern, separation point, drag coefficient and Strouhal number. This could be a possible explanation for the observed improvement. Despite this improvement, the current model does not provide any improvement in the prediction of the turbine wake asymmetry. A thorough discussion on the inability of the investigated models to predict this is given in Ref. [51].

3. Turbine power loss due to the presence of the shaft

3.1. Impact of shaft-to-diameter ratio

Simulations are performed for different shaft-to-diameter ratios (\( \delta \)) to quantify the power loss due to the presence of the shaft. The turbine instantaneous moment coefficient \( (C_{m}) \) versus azimuth for the last turbine revolution for various \( \delta \) from 2% to 16%, which are relevant ratios for urban-scale VAWTs, are compared in Fig. 6. The results of the case without shaft is also shown in this figure. For the case with no shaft, the turbine power and thrust coefficients \( (C_{P} \) and \( C_{T} \)) are significantly reduced. The turbine power and thrust coefficients \( (C_{P} \) and \( C_{T} \)) are significantly reduced. The turbine power and thrust coefficients \( (C_{P} \) and \( C_{T} \)) are significantly reduced. The turbine power and thrust coefficients \( (C_{P} \) and \( C_{T} \)) are significantly reduced.
shaft, $C_m$ stays almost constant on the downwind side of the revolution ($225° < \theta < 315°$). However, the presence of a shaft with $\delta \leq 4\%$ results in a drop in $C_m$ near $\theta = 270°$, as a result of the blade passing through the wake of the shaft. Note that strong interaction of the blade with the vortices shed by the shaft is hardly observed in the downwind quasitile. In contrast, for $\delta > 4\%$, there is a clear effect of vortex shedding from the shaft on the $C_m$ curve. In this case, a drop in $C_m$ near $\theta = 270°$ is observed while simultaneously an increase at neighboring values of $\theta$ is found. The change in turbine $C_p$ for different values of $\delta$ is shown in Fig. 6b. The turbine power loss (regarding reduction in $C_p$) due to the presence of the shaft asymptotically increases from 0 to 5.5% with increasing $\delta$ from 0 (no shaft) to 16%.

The decrease in $C_p$ with increasing $\delta$ can be associated with the increase in the width of the shaft wake at the downstream location where the blades pass ($x/d = 0.5$), as shown in Fig. 7. The increase in the width of the shaft wake occurs together with the decrease in the mean streamwise velocity. The time-averaged (over 10 turbine revolutions) normalized streamwise velocity at $x/d = 0.5$, averaged along the lateral line $-0.3 \leq y/d \leq 0.3$, decreases from 0.44 to 0.40 as $\delta$ increases from 2% to 12%. The asymptotic trend observed for $C_p$ (see Fig. 6b) is because at high $\delta$ the complete blade trajectory on the downwind side intersects the shaft wake and the contribution of this quartile to the overall turbine performance, therefore, becomes negligible.

Fig. 8 presents contours of dimensionless out-of-plane vorticity ($\omega/d_{c}/U_{\infty}$) in the wake of the shaft for different $\delta$. It can be seen that the strength of the interactions between the blades and the shed vortices from the shaft increases non-linearly with increasing $\delta$. As $\delta$ increases, the shaft wake interactions with the blade become stronger due to the larger size of the vortices passing through the blade trajectory, but also the smaller dimensionless streamwise distance ($x/d_{c}$) from the shaft to the blade trajectory. This is consistent with the trend observed for $C_m$ in Fig. 6a. Moreover, Fig. 8 indicates that separation on the leeward side ($y/d_{c} < 0$) of the rotating shaft happens later than on the windward side ($y/d_{c} > 0$) which is expected for a shaft rotating in counterclockwise direction. This delay in separation on the leeward side is due to the fuller velocity profile in the boundary layer commonly observed for a downstream moving wall [70].

### 3.2. Sensitivity analysis: impact of operational and geometrical parameters

A sensitivity analysis is performed for other operational and geometrical conditions to further generalize the turbine power loss due to the presence of the shaft. The impact of several operational parameters including inlet turbulence intensity (TI), chord-based Reynolds number ($Re_{c}$) and tip speed ratio ($\lambda$), and geometrical parameters including solidity ($\sigma$) and number of blades ($n$) are investigated. Some of the main parameters of the cases that are investigated are given in Table 6.

The effort is made to investigate the effect of each parameter while the others are kept constant. Note that $\lambda$ is normally changed by either altering $\Omega$ or $U_{\infty}$. The former is employed in the current study, which means that while changing $\lambda$ (for a given $U_{\infty}$), $Re_c$ is also altered. Similarly, solidity is typically modified by changing the blade chord length which means that (for a given $U_{\infty}$ and $\Omega$), $Re_c$ also changes. The effect of number of blades is studied at constant blade chord length, which inevitably changes solidity as well.

Fig. 9a shows the impact of $Re_c$ ($132 \times 10^3 \leq Re_c \leq 352 \times 10^3$) on the turbine power loss when the shaft is present ($\delta = 4\%$). This range corresponds to shaft diameter-based Reynolds number $29 \times 10^4 \leq Re_c \leq 76 \times 10^4$. $Re_c$ is changed by increasing the freestream velocity from 6.98 to 18.6 m/s while $\Omega$ is kept constant. The trend shows that by increasing $Re_c$ from $132 \times 10^3$ to $352 \times 10^3$, the negative effect of the shaft on the turbine performance reduces as $|\Delta C_p|$ decreases from 2.5% to 1.6%. The aforementioned operating regime of

### Table 5

Average deviation of the time-averaged (over 10 turbine revolutions) normalized velocity from experiment [50].

<table>
<thead>
<tr>
<th>$x/R$</th>
<th>2.0</th>
<th>2.5</th>
<th>3.0</th>
<th>3.5</th>
<th>4.0</th>
</tr>
</thead>
<tbody>
<tr>
<td>$u/U_{\infty}$</td>
<td>k-omega SST int.</td>
<td>6.7</td>
<td>7.9</td>
<td>9.2</td>
<td>9.7</td>
</tr>
<tr>
<td>transition SST</td>
<td>8.6</td>
<td>10.0</td>
<td>11.8</td>
<td>12.6</td>
<td>16.4</td>
</tr>
<tr>
<td>$v/U_{\infty}$</td>
<td>k-omega SST int.</td>
<td>2.5</td>
<td>2.3</td>
<td>2.5</td>
<td>2.1</td>
</tr>
<tr>
<td>transition SST</td>
<td>2.5</td>
<td>2.3</td>
<td>2.2</td>
<td>2.3</td>
<td>2.9</td>
</tr>
</tbody>
</table>

Fig. 4. Computational grid: (a) rotating core; (b) airfoil; (c) airfoil trailing edge; (d) airfoil leading edge.
The shaft corresponds to the subcritical regime for cylinders \[39\]. The studies on cylinders have shown \[39,40\] that within this regime increasing Re will shift the flow on cylinder towards critical, reduce the drag coefficient, delay the flow separation and abate the shaft wake width which eventually will enhance the turbine CP. Further explanations on the effect of delay in separation on the shaft and the improvement in turbine aerodynamic performance are provided in Section 4.1.

The impact of inlet turbulence intensity on the turbine power loss due to the shaft \( \Delta C_P (\delta = 4\%) \) is shown in Fig. 9b. In this case, \( 0\% \leq TI \leq 10\% \), which corresponds to incident turbulence intensity \[51,54,55\] ranging from 0 to 7.9\%. It can be seen that changing the inlet TI from 0 to 10\% reduces \( |\Delta C_P| \) from 2.6 to 1.9\%. The increase in TI leads to the promotion of laminar-to-turbulent transition in the boundary layer on the shaft and consequently delays the separation and reduces the shaft wake, which benefits the turbine CP (see Section 4.1 for further explanation).

Fig. 9c depicts the impact of tip speed ratio \( 3.5 \leq \lambda \leq 5.5 \) on the turbine power loss due to the shaft \( \Delta C_P (\delta = 4\%) \). The tip speed ratio is altered by changing \( \Omega \) at constant \( U_\infty \) which inevitably also changes \( Re_c \) between \( 139 \times 10^3 \) and \( 214 \times 10^3 \). Maximum turbine power loss due to the presence of the shaft occurs at the optimal tip speed ratio where the maximum CP is generated. This shows that at the tip speed ratio where the turbine works most optimally the shaft creates the maximum loss of 2.3\%. This value reduces to 1.8\% and 1.2\% at a tip speed ratio of 3.5 and 5.5, respectively. The smaller \( |\Delta C_P| \) at \( \lambda = 5.5 \), compared to \( \lambda = 3.5 \), could be associated to higher \( Re_c \) at this \( \lambda \).

Fig. 9d illustrates the impact of solidity \( 0.09 \leq \sigma \leq 0.18 \) on the turbine power loss due to the shaft \( \Delta C_P (\delta = 4\%) \). As the solidity increases, the relative size of the blade chord length to shaft diameter (and consequently shaft wake) increases. This results in the fact that when blades pass through the shaft wake, they are to a less extent affected by the shaft wake. Therefore, increasing the solidity and blade chord length reduce \( |\Delta C_P| \).

Fig. 10 shows the impact of number of blades on the turbine power loss due to the shaft \( \Delta C_P (\delta = 4\%) \) for a two- and three-bladed turbine with \( \delta = 4\% \) and 8\%. The blade chord length is kept constant. This makes the solidity change between 0.12 and 0.18. It can be seen that increasing the number of blades has a similar impact as increasing the
solidity (see Fig. 9d). $|\Delta C_p|$ reduces from 2.3% and 1.8% for $\delta = 4$ and from 3.9% to 1.2% for $\delta = 8$%. Increasing the number of blades results in more energy extraction in the fore half of the turbine, therefore, less energy can be potentially extracted in the aft half. This already results in less contribution of this region in the turbine total power production. As the power loss due to the shaft occurs in the aft half, less contribution of this region leads to a decrease in $|\Delta C_p|$. The reduction in $|\Delta C_p|$ by increasing number of blades becomes stronger when the power loss due to the shaft is also higher ($\delta = 8$%).

4. Turbine power improvement

4.1. Impact of shaft surface roughness

4.1.1. Average loading

The variations of $C_p$ and $C_T$ for the last turbine revolution with dimensionless equivalent surface roughness height $k_s/d_s$ of the shaft are shown in Fig. 11a. It can be seen that adding roughness can increase $C_p$, with an optimal increase of 1.7% for $k_s/d_s = 0.08$. Since the power loss due to the presence of the turbine shaft ($\delta = 4$%) is 2.4% (Fig. 6b), no less than 69% of the power loss due to the presence of the shaft is regained. Higher $k_s/d_s$ values will not further improve the performance of the turbine, as the flow on the cylinder most probably shifts to the supercritical regime [40]. A similar trend is observed for the thrust force (Fig. 11a), with an increase in $C_T$ of 1.2%.

The time-averaged (over the last turbine revolution) drag and lift coefficients ($C_d$ and $C_l$) of the shaft are shown in Fig. 11b. The optimum power coefficient (Fig. 11a) corresponds to the roughness height in
which the minimum shaft $C_d$ occurs. This implies that the power improvement of the turbine is partly a result of the reduced drag on the shaft itself. Furthermore, the variations of $C_d$ as a function of $k_r/d_r$ typically signals a shift from the subcritical to the critical regime for flow over cylinders [40]. In other words, increasing the roughness height will result in a reduction in the critical Reynolds number $Re_{crit}$ and eventually, for an optimal roughness height, it will equal $Re_b$. An empirical correlation between $k_r/d_r$ and $Re_{crit}$ was deduced by Buresti [52] and Fage and Warsap [53] (see Eq. (1)). Based on this correlation, the identified optimal $k_r/d_r = 0.08$ yields $Re_{crit} \approx 21,000$, which is very close to the $Re_b$ of the shaft provided in Table 4 ($Re_b = 26,000$). This further confirms that the observed reduction in shaft $C_d$ and consequent turbine power recovery is due to the shift of the flow over the shaft from subcritical to critical.

Note: by changing $\lambda$ and $\sigma$, $Re_c$ is also altered. Also by changing $n$, $\sigma$ is also changed.

Table 6
Some of the main parameters that are studied in the sensitivity analysis. Blade diameter is 1 m for all cases.

<table>
<thead>
<tr>
<th>Parameter to study</th>
<th>$n$</th>
<th>$\delta$ [%]</th>
<th>$\sigma$ [cm]</th>
<th>$\lambda$</th>
<th>$U_\infty$ [m/s]</th>
<th>$\Omega$ [rad/s]</th>
<th>$Re_b (\times 10^3)$</th>
<th>TI [%]</th>
</tr>
</thead>
<tbody>
<tr>
<td>$Re_b$</td>
<td>2</td>
<td>0, 4</td>
<td>0.12</td>
<td>6.0</td>
<td>4.5</td>
<td>6.98</td>
<td>62.77</td>
<td>132</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>9.30</td>
<td>83.70</td>
<td>176</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>11.62</td>
<td>104.62</td>
<td>220</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>13.95</td>
<td>125.55</td>
<td>264</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>18.60</td>
<td>167.40</td>
<td>352</td>
</tr>
<tr>
<td>$TI$</td>
<td>2</td>
<td>0, 4</td>
<td>0.12</td>
<td>6.0</td>
<td>4.5</td>
<td>9.30</td>
<td>83.70</td>
<td>176</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>0</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>10</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>20</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>30</td>
</tr>
</tbody>
</table>

Fig. 9. The variations of turbine power loss due to the presence of the shaft for different (a) chord-based Reynolds number, (b) inlet turbulence intensity, (c) tip speed ratio and (d) solidity. $\delta = 4\%$ for all cases.
4.1.4. Flow field

Vorticity contours with superimposed streamlines for the smooth and rough shaft are compared in Fig. 15. It can be seen that for the rough shaft the separation on both upper and lower surfaces of the shaft is delayed (see Table 7), although the delay is much more significant on the upper surfaces where the upstream-moving wall is more prone to separation. Here, the roughness in the boundary layer is shown to be more effective in increasing the resistance of the boundary layer to separation by adding extra momentum and promoting the laminar-to-turbulent transition. This also reduces the lateral asymmetry in the separation point locations on the upper and lower shaft surfaces as well as the shaft wake asymmetry.

In order to further elaborate on the delay in separation for the rough shaft, the profiles of time-averaged (over 10 turbine revolutions) normalized tangential velocity \( V_{\phi}/U_{\infty} \) on the upper surfaces of the smooth and rough shafts are compared at circumferential angles \( 70° \leq \phi \leq 125° \), where separation is expected to happen in both cases (see Fig. 16). It is observed that the boundary layer is still attached for both shafts at \( \phi = 70° \) while reverse flow (\( V_{\phi}/U_{\infty} < 0 \)) is already present for the smooth shaft at \( \phi = 80° \). The separation grows for the smooth shaft with increasing \( \phi \) while the flow stays attached up to \( \phi = 110° \) for the rough shaft. At this point, reverse flow is also observed for the rough shaft, and the separation (regarding the radial distance of the location of zero velocity from the shaft wall) grows up to \( \phi = 125° \).

The observed delay in the separation over the rough shaft means that the amount of separated flow over the shaft has significantly decreased, which is the reason for the reduction in the amplitude of fluctuations of...
instantaneous $C_l$ and $C_d$ shown in Fig. 13.

A comparison of time-averaged (over the last turbine revolution) distributions of pressure coefficient ($C_{op}$) over the smooth and rough shafts ($k_s/d_s = 0.08$) is given in Fig. 17a. It is shown that the pressure difference between fore ($0° \leq \varphi < 90°$ and $270° < \varphi \leq 360°$) and aft ($90° \leq \varphi \leq 270°$) halves of the shaft reduces significantly for the rough shaft, which is consistent with the substantial reduction in $C_d$ observed in Fig. 13b. The asymmetry in the pressure distribution of both cases is a result of the counter-clockwise rotation of the shaft. The upper shaft surface moves against the freestream direction, leading to a lower relative velocity and a higher pressure at $\varphi \approx 90°$. The reverse occurs on the lower surface.

Skin friction coefficients $C_f$ for the smooth and rough shaft ($k_s/d_s = 0.08$) are compared in Fig. 17b. Note that zero-crossings in this plot can be used to detect flow separation. The comparison further confirms the delay in separation for the rough shaft on both upper and lower surfaces. For example, on the upper surface, the zero-crossing for the smooth shaft happens well before $\varphi = 90°$ while this occurs beyond $\varphi = 90°$ for the rough shaft.

The time-averaged (over 10 turbine revolutions) normalized streamwise velocity along the symmetry line $y/d = 1$ and $y/d_s = 0.5$ is shown in Fig. 18. The higher mean velocity at the location where blades pass ($x/d = 0.5$) is apparent for the rough shaft. This is also the case for other lateral positions (not shown in this figure). The time-averaged (over 10 turbine revolutions)
The normalized streamwise velocity along the lateral line $-2 \leq y/d_s \leq 2$ in different downstream locations $0.5 \leq x/d_s \leq 12.5$ ($0.02 \leq x/d \leq 0.5$) for the smooth and rough shafts is shown in Fig. 19. The smaller amount of separation and narrower shaft wake for the rough shaft can be observed. The recovery in the instantaneous moment coefficient of the turbine observed in Fig. 12 and the improvement in turbine $C_P$ shown in Fig. 11a for the rough shaft are a direct result of these observations. In addition, the reduction in the shaft wake width results in a smaller length scale of vortical structures in the wake of the shaft with a higher vortex shedding frequency $f$. This explains the observed increase in $St$ for the rough shaft shown in Fig. 14.

4.2. Impact of shaft-to-turbine rotational speed ratio

The impact of the shaft-to-turbine rotational speed ratio $\eta$, given by Eq. (3), on the performance of the turbine for a rough shaft with the optimal roughness height identified in Section 4.1 ($k_s/d = 0.08$) is studied for $\eta$ ranging from 0 to 1. This is important as it can determine to which extent results obtained in Section 4.1 ($\eta = 1$) depend on $\eta$. Note that the value of $\eta = 0$ corresponds to a stationary shaft while $\eta = 1$ means that the shaft is rotating at the same speed and direction as the turbine. Fig. 20 shows a negligible difference on $C_m$ and $C_P$ ($< 0.2\%$) within the studied range. Therefore, the identified aerodynamic performance of the turbine in Section 4.1 is not significantly affected by $\eta$.

5. Conclusions

URANS calculations are performed to study the effect of the shaft on the aerodynamic performance of urban-scale VAWTs. Turbulence is modeled using the 3-equation $k$-$\omega$ SST model with the intermittency transition submodel. The evaluations are based on two sets of validation studies for VAWTs. Furthermore, as the focus of the paper is on the effect of the shaft on turbine performance, separate validation studies are performed for flow over smooth and rough cylinders. Investigated parameters are the shaft-to-turbine diameter ratio $\delta$, the dimensionless shaft surface roughness $k_s/d_s$ and the shaft-to-turbine rotational speed ratio $\eta$. In addition, the impact of several operational and geometrical parameters on the turbine power loss due to the presence of the shaft is investigated. The following conclusions can be drawn:

- The power loss of the turbine regarding reduction in $C_P$ increases asymptotically from 0 to 5.5% for $\delta$ from 0 to 16%. This is due to the larger low-velocity wake region behind the shaft, where the blades pass through on the downwind side of the turbine. For large values of $\delta$ (> 4%), the interaction between the vortices shed from the shaft, and the blades significantly increases.
- Increasing chord-based Reynolds number, inlet turbulence intensity, solidity and number of blades are found to decrease the turbine...
The rotating shaft for an urban-scale VAWT operates in the sub-critical regime with a $R_e$ which is one order of magnitude lower than that of the blade. This results in a high drag coefficient $C_d$ for the shaft.

The addition of surface roughness is systematically investigated. The following conclusions are made:

- The flow over the shaft is shifted from subcritical to critical. Several parameters are employed to establish that this change in flow regime occurs between the smooth shaft and the rough shaft (with optimal roughness height $k_s/d_s = 0.08$):
  a. A delay in separation point over both the upper (from $\phi = 74^\circ$ to $110^\circ$) and lower (from $\phi = 223^\circ$ to $244^\circ$) surfaces of the turbine shaft and a consequent notable reduction in the shaft wake width;
  b. A jump in Strouhal number from 0.19 to 0.23;
  c. A significant reduction in average shaft drag.
- Because of the shift from subcritical to critical, the separation on the shaft is delayed and, as a result, the shaft wake width decreases and a larger mean velocity is present at the downstream turbine blades’ trajectory. This means that the blades pass through a smaller wake region with higher mean velocity and consequently turbine performance is improved.
- An optimal shaft surface equivalent sand-grain roughness height of $k_s/d_s = 0.08$ is found to improve the turbine $C_p$ by over 1.7%, leading to a 69% recovery of the power loss due to the presence of the shaft $|\Delta C_p|$. The minimum $|\Delta C_p|$ occurs at the optimum tip speed ratio, where $C_p$ is maximum.
- Within the range of parameters studied, the impact of the shaft-to-turbine rotational speed ratio ($\eta$) on the performance of the turbine is negligible.

It should be noted that the focus of the current study is to quantify
the turbine power loss associated with the shaft for an urban-scale VAWT and to show that employment of an optimal roughness height can significantly recover the identified power loss and improve turbine performance. From a practical point of view, adding roughness to the shaft is a simple and inexpensive method. Therefore, investigation of the performance of the turbine with a rough shaft at other tip speed ratios, solidities and blade shapes can be very intriguing for future research. Furthermore, three-dimensional CFD calculations and the application of more advanced hybrid RANS-LES methods can provide further details of the 3D and unsteady nature of the interactions between the vortices shed by the shaft and the blades passing downstream.

Acknowledgements

The authors would like to acknowledge support from the European Commission’s Framework Program Horizon 2020, through the Marie Curie Innovative Training Network (ITN) AEOLUS4FUTURE - Efficient harvesting of the wind energy (H2020-MSCA-ITN-2014: Grant agreement no. 643167) and the TU1304 COST ACTION “WINERCOST”. The authors gratefully acknowledge the partnership with ANSYS CFD. This work was sponsored by NWO Exacte Wetenschappen (Physical Sciences) for the use of supercomputer facilities, with financial support from the Nederlandse Organisatie voor Wetenschappelijk Onderzoek (Netherlands Organization for Scientific Research, NWO). The 3rd author, Hamid Montazeri, is currently a postdoctoral fellow of the Research Foundation – Flanders (FWO) and is grateful for its financial support (project FWO 12M5316N).

References

[37] Blocken B, Cardinael J, Staes Pouliot T. CFD evaluation of wind speed conditions in


