3-D BLADE RESOLVED CFD PERFORMANCE ANALYSIS OF DUCTED WIND TURBINES

R. Bontempo - E.M. Di Marzo - M. Manna

Dipartimento di Ingegneria Industriale, Università degli Studi di Napoli Federico II, via Claudio 21, 80125 Naples, Italy

ABSTRACT
Ducted wind turbines are very promising wind energy concentrator-systems for small-scale applications exhibiting an appreciable power density increase compared with their open rotor competitors. The aim of this work is to build up a robust and automated methodology for the 3D blade-resolved performance analysis of these devices in view of their CFD-aided optimal design. The reliability of this procedure, based on an in-house Python module for the generation of the structured multi-block 3D mesh, is verified by comparing the obtained numerical results with available experimental data. The developed methodology is employed to spotlight the differences in terms of extracted power between open and ducted rotors and to investigate some typically-disregarded aspects playing a key role in the design and analysis of ducted wind turbines. Specifically, it is shown that the shrouding of an existing open rotor leads to a sub-optimal turbine operating condition as well as to a significantly change in the local features of the tip vortex and associated losses.

KEYWORDS
Ducted wind turbines, blade resolved simulations, CFD

NOMENCLATURE

\[ D \] \quad \text{Rotor diameter} \\
\[ D_{ex} \] \quad \text{Duct exit diameter} \\
\[ C_p = \frac{p}{\frac{1}{2} \rho V_\infty^2} \] \quad \text{Pressure coefficient} \\
\[ C_P = \frac{\frac{1}{2} \rho V_\infty^2 \frac{V_T^2}{4}}{p} \] \quad \text{Power coefficient} \\
\[ C_{P,ex} = \frac{\frac{1}{2} \rho V_\infty^2 \frac{V_T^2}{4}}{\frac{1}{2} \rho V_\infty^2 \frac{V_T^2}{4}} \] \quad \text{Power coefficient based on duct exit area} \\
\[ C_T = \frac{\frac{1}{2} \rho V_\infty^2 \frac{V_T^2}{4}}{T} \] \quad \text{Thrust coefficient} \\
\[ V_\infty \] \quad \text{Free-stream velocity} \\
\[ z, r, \theta \] \quad \text{axial, radial and tangential direction}

INTRODUCTION
The tendency towards clean energy production has pushed the research in the field of renewable energies. In this context, wind energy is very promising since it is 100% pollution free, although its power density is moderate. Accordingly, the rotor size has constantly increased during the years to satisfy the growing energy demand. However, following recent trends aimed at complementing the current centralized generation paradigms with distributed systems where
electricity is produced next to its end-users, research activities are progressively focusing on
small-sized wind turbines, too. In this context, recent developments have showed that Ducted
Wind Turbines (DWTs) are very promising at increasing the device power density. Indeed, for a
given rotor diameter, the sectional circulation of the duct increases the mass flow swallowed by
the rotor, thus increasing the delivered power. Alternatively, this solution allows to reduce the
device size keeping unaltered the energy extraction. Moreover, due to the lower cut-in speed,
DWTs are well suited for urban environment installations (Vita et al., 2020a,b), and they are
also characterized by a lower level of noise, tip losses and sensitivity to yaw angles. However,
costs and structural requirements for the installation are higher.

In order to understand their operating principles, without carrying out expensive experi-
mental campaigns or numerical simulations, several theoretical models (Bontempo and Manna,
2020b) have been developed during the years as those of Lilley and Rainbird (1956), Foreman
et al. (1978), van Bussel (1999), Lawn (2003), and Sorribes-Palmer et al. (2017), to name a
few. In these models, the flow is assumed to be steady, incompressible and axisymmetric, while
the rotor is replaced by an actuator disk, i.e. a disk of infinitesimal thickness which induces
on the flow the same effects of the rotor. Recently, Bontempo and Manna (2020a) developed
a theoretical model based on the Axial Momentum Theory (AMT) which takes into account
the presence of the tip gap as well as the 2D features of the flow. As well known, the main
shortcoming of the AMT relies in its inability to account for the mutual interaction between
the duct and rotor. However, the authors showed that the AMT is very accurate once that correct
parameters are given as input. These values were obtained by a free-wake ring-vortex actuator
disk model (Bontempo and Manna, 2018, 2019). They also showed that the duct-thrust ratio
($\tau_D$) plays a crucial role in the performance augmentation since a positive $\tau_D$ value leads to
higher extracted power than the corresponding Open rotor Wind Turbine (OWT). In particular,
the upper duct side and the diffusion region may determine the excess of the Betz limit.

However, more advanced models also exist. van Dorst (2011) combined a constant-strength
panel approach, for the hub and duct, with a vortex-line representation of the rotor whose
aerodynamic-loads are evaluated via the classical Blade-Element-Theory. He founded out dis-
crepancies between the model and the experiments within 15%. Bontempo and Manna (2013,
2014) extended the model developed by Conway (1995), deriving a semi-analytical approach
for solving the flow around a ducted rotor showing an excellent agreement with the CFD-
Actuator Disk (CFD-AD) at a fraction of its cost.

The aforementioned CFD-AD method couples a standard CFD methodology for the flow-
field solution with the representation of the rotor through an actuator disk. Specifically, in this
approach, which is widely spread in the analysis and design of DWTs, the effect of the rotor is
simulated by introducing a set of body forces in the momentum equations, which act as source
terms in the disk region, a small volume of finite thickness. Dighe et al. (2017, 2019) used
this methodology to analyze the performance of DWT, to investigate the effect of the Gurney
flap and to perform an optimization process which led to a highly cambered duct section. They
found out that any gain in the velocity at the rotor plane increases the power extraction even if
all analysed configurations experienced flow-separation on the upper side of the duct.

As in classical turbomachinery problems (Bernardini et al., 2011), blade-resolved 3D anal-
yses have also been proposed to investigate some crucial aspects of the flow field. Cosioiu et al.
(2011) compared the performance of an open and ducted wind turbine, showing an higher $C_P$
for the former equal to 2.09 times the Betz limit. Wang et al. (2012) designed and analyzed the
performance of a multi-blade DWT obtaining a $C_P$ two times higher than the corresponding
OWT. Aranake et al. (2013, 2015) enclosed an existing open rotor (the NREL Phase VI) in a duct made up with the Selig S1223 profile as cross section. Their 3D-CFD analysis led to a maximum $C_P$ equal to 1.13. They also noticed a strong interaction between the helical wake and the boundary layer developing over duct internal wall, where the high-momentum swirling wake-flow re-energizes the low momentum fluid delaying the separation from the duct surface. Successively, Aranake and Duraisamy (2016) used a CFD-AD method coupled with the inverse Blade Element Theory to design a DWT obtaining a $C_P$ of 1.32. However, when the same geometry was investigated with a 3D blade resolved solver, it resulted in a $C_P$ of 1.57, thus demonstrating that some 3D effects not taken into account in the axisymmetric CFD-AD analysis turned out to be essential. A detailed study of the shape of the diffuser installed around a commercial OWT has been conducted by Jafari and Kosasih (2014). Their simulations showed that better performance are achieved increasing the outlet-to-inlet area-ratio and increasing the length of the diffuser. The only limitation for the value of this geometrical ratio is given by the separation onset in the lower surface of the diffuser; similar results have been obtained by Noorollahi et al. (2019). Avallone et al. (2020) used the Lattice-Boltzmann method to perform the unsteady analysis of a DWT. Their simulations showed that the tip gap flow helps to delay the separation on the duct suction side, resulting in a strong positive impact on the overall performance. They suggested that higher value of the tip gap could be beneficial for higher energy extraction. In a further study, Dighe et al. (2019, 2020) used the same methodology to investigate the effect of the inflow angle for a non optimized and optimized turbine. They showed that the ducted turbine turns out to be less sensitive to the inlet angle when compared with the bare turbine.

From the above literature review, it clearly appears that the flow field through a DWT is more complicated than that of a bare turbine because of several phenomena. For example, the strong interaction between the duct and the rotor influences the ingested mass flow and the duct operating condition. Moreover, the tip vortex forming in a stronger adverse-pressure-gradient, interacts with the duct boundary layer and then mixes out with the wake coming from the duct trailing edge. These flow features and their mutual dependency can be properly taken into account only via a 3D blade-resolved model which is then more advisable to adopt both for the analysis and the design of this kind of wind-concentrator systems. In this context, the present paper has two aims. Firstly, it presents an effective mesh topology as well as a reliable and automated grid generation procedure to be used in an optimization algorithm for the design of DWTs. Secondly, the proposed methodology is also employed to shed some light on a few typically-disregarded aspects related to the design and analysis of DWTs. For example, many DWT designs rely on the shrouding of an existing open rotor, a practice which ignores the flow-variation induced by the duct at the turbine inlet, thus leading to a sub-optimal rotor performance. Moreover, some tools valid for the open rotor configuration, e.g. the Glauert optimal rotor and the tip loss models (Sørensen, 2015), are bluntly applied to the design and analysis of the ducted case without any specific modification.

The first part is dedicated to the description of the automated topology and grid generation process. Then, to test the reliability of the automated strategy as well as the stability and convergence of the associated simulations, the obtained results are compared with experimental data in terms of global performance coefficient. Finally, the main differences in the operating conditions as well as in the local flow features between an open and ducted wind turbines are investigated.
METHODOLOGY

Ducted rotors mesh generation is one of the most complex items to be handled when blade resolved simulations with controlled accuracy are looked for. This is particularly true for high Reynolds number flows requiring highly stretched, anisotropic grids which become difficult to build when large scale disparities are present in the geometry. In this work, a structured multi-block mesh with conformal interfaces is generated through a fully-automated procedure by a Python code. The device geometric features, either pre-processed off-line or generated through a set of design variables parametrizing the components, are provided as input to the routine. Then the blocks of the mesh are created, and, finally, an elliptic grid generator is employed to build the whole mesh. In order to reduce the computational cost, only one pitch is considered, so that the final grid is a cylindrical sector whose radial extent is $20 \times R$, while its axial length is $30 \times R$. The domain is divided into 2 parts, i.e., the rotor and the external region. In the rotor zone, several 2D blade-to-blade meshes (consisting of four H blocks connected to an O-grid wrapped around the blade-section) are generated and, successively, stacked in the spanwise direction to obtain the 3D rotor grid. The tip gap zone is discretized by extruding the rotor tip section mesh towards the duct inner surface. In the external domain, a 2D mesh in a meridional plane is built by surrounding the duct with a C-block, while H blocks are used anywhere else. The 3D mesh is created by rotating the 2D grid around the axis. To further reduce the overall computational cost, the size of the external-domain cells is progressively decreased moving towards the outer boundaries (see Figure 1a). A fully conformal interface is enforced between the rotor and the external domains, while a butterfly topology is used near the rotational axis in order to increase the local grid quality. The width of the mesh-layer adjacent to the solid boundaries of the hub, blade and duct is evaluated setting a $y^+ = 1$ target-value, estimated using approximate relations for fully turbulent boundary layers. Regarding the boundary conditions, a uniform velocity is imposed at the inlet while the static pressure is fixed at the outlet. The upper face of the cylindrical domain is modelled as an inviscid wall, while periodic conditions are used on the two lateral surfaces (see Fig. 1a). The blade and the spinner are rotating walls, whereas the nacelle and the duct are fixed.

The mathematical model relies on the Reynolds-Averaged Navier-Stokes equations which are numerically solved by the commercial software ANSYS Fluent, using a classical finite volume cell-centered approach. The closure of the problem is made through the Shear Stress Transport turbulence model (Menter et al., 2003). The equations are formulated in the rotating frame of reference but solved for the absolute velocity components for stability reasons. The
Weiss and Smith (1995) low-Mach preconditioner is used to increase the efficiency of the compressible time-marching algorithm. The viscous fluxes are discretized by a pure central scheme, while a central scheme with artificial dissipation is used for the inviscid fluxes.

RESULTS

The pre-processing of the parametrized geometry (rotor, duct and hub), the domain partitioning and the 3D mesh generation are carried out in a fully automated manner so that they can be readily integrated in an optimization scheme for the design of DWTs. The high level of automatization required by the design systems based on optimal control theory offers no opportunities to interfere with the analysis procedure which needs to be robust. To test the reliability of this automated grid-generation strategy as well as the stability and convergence of the associated numerical simulations, the proposed procedure is used to evaluate the performance coefficients of the small-sized urban windmill DonQi developed and tested at the Technical University of Delft (van Dorst, 2011). The rotor diameter $D$ of this device is equal to 1.5m, the blade sections are standard NACA 2207 profiles, the twist angle has a classical decreasing trend along the spanwise direction, while the chord is nearly uniform all along the blade. The axial length of the duct and the tip-gap are 66% and 1% of the rotor diameter, respectively. The experimental campaign is carried out in an open-jet facility with a variable rotational speed of the rotor. For different values of the free-stream velocity, Fig. 2 shows the maximum mechanical power obtained sweeping each iso-rpm curves (maximum power coefficient point) along with the associated numerical results. Although, it clearly appears that the power is slightly under-predicted, the overall agreement can be considered satisfactory, also taking into account the unknown uncertainties of the experimental data (operating conditions and power).

Comparison between open and ducted rotor

As mentioned in the introduction, the proposed automated procedure for the blade-resolved simulation of DWTs is also employed to shed some light on a few typically-disregarded aspects related to the design and analysis of these devices. For example, many DWT designs rely on the shrouding of an existing open rotor, a practice which ignores the flow-variation induced by the
duct at the turbine inlet, thus leading to a sub-optimal rotor performance. Moreover, some tools valid for the open rotor configuration, e.g. the Glauert optimal rotor and the tip loss models (Sørensen, 2015), are applied to the design and analysis of the ducted case without any specific modification which is clearly requested on account of the differences characterizing the open rotor and ducted tip flows. Therefore, this subsection deals with the comparison between the performance and the local-flow features of a well-known OWT, i.e. the NREL Phase VI (Hand et al., 2001), and those of a DWT obtained by enclosing this rotor in an annular duct. Since this comparison is simply aimed at showing the main difference induced by the duct presence on the flow, its shape is not optimized and, therefore, the resulting DWT is not expected to exhibit particularly brilliant performance. The rotor housing is summarized in Fig. 3. The duct is generated by the azimuthal revolution of a NACA 5415 profile, while the hub is divided into the spinner (made of an elliptic ogive and a cylinder) and a parabolic-shaped nacelle. The relative position between the duct, the hub and the rotor is determined using the blade stacking line as reference. Specifically, the midpoint of the hub cylindrical-part as well as the duct throat are located at the same axial station of the stacking line. To limit the computational effort, the extent of the domain is reduced by increasing the blade count from 2 (as in the classical Phase VI rotor) to 3. The Reynolds number based on the duct chord is $Re_c = 2.58 \cdot 10^6$ and the TSR = 6.736. As detailed in Tab. 1, the DWT exhibits higher performance compared to the corresponding OWT with a $C_P$ gain of 31%, even though it does not beat the Betz-Joukowsky limit (0.593). As previously stressed, this low $C_P$ value is not surprising because the rotor and

<table>
<thead>
<tr>
<th>Case</th>
<th>$\dot{m}$ [kg/s]</th>
<th>$C_P$</th>
<th>$C_{P,ex}$</th>
<th>$C_{T_R}$</th>
<th>$C_{T_D}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>OWT</td>
<td>540.66</td>
<td>0.38</td>
<td>-</td>
<td>0.62</td>
<td>-</td>
</tr>
<tr>
<td>DWT</td>
<td>567.54</td>
<td>0.50</td>
<td>0.348</td>
<td>0.74</td>
<td>0.146</td>
</tr>
</tbody>
</table>

Table 1: Performance parameters of the open and ducted wind turbines

the duct are not optimized to work in a coupled manner. This can also be understood inspecting Fig. 4a which reports the azimuthally-averaged pressure-coefficient distribution along the duct.

Figure 3: Ducted Phase VI dimensions expressed as percentage of the rotor diameter.
Figure 4: Azimuthally-averaged quantities for the ducted NREL Phase VI turbine. (a) Duct pressure coefficient $C_p$ versus non dimensional chord. (b) Axial velocity and relative flow angle variation $\Delta \beta$ (all quantities are evaluated at an axial station located 0.02D ahead of the stacking line).

The figure clearly shows that the shroud is not working at the optimal condition, since its front-stagnation-point is far from the leading edge. This result is perfectly in line with the findings of Bontempo and Manna (2020a) who showed that for high values of the rotor thrust coefficient ($C_{TR} = 0.74$ as reported in Tab. 1) the front-stagnation-point tends to shift towards the inner side of the duct. The increase in the extracted power obtained by ducting the turbine is due to the raise of the mass-flow $\dot{m}$ sucked by the rotor which, as detailed in Tab. 1, goes from 540.66 to 567.54 kg/s. This phenomenon can also be observed in Fig. 4b presenting the radial distribution of the axial velocity $v_z$ at the rotor inlet. As it can be easily inferred, the duct alters the rotor inflow conditions, thus inducing a change in the incidence angle all along the blade span. To better clarify this aspect, Fig. 4b also shows the variation in the relative flow-angle $\Delta \beta = \beta_{DWT} - \beta_{OWT}$ which, since the two rotors have the same rotational speed, leads to a modification of the rotor local incidence. Consequently, the axial and tangential blade forces are also mutated, as shown in Fig. 5. The increase in the axial-force linear-density $\Delta T/\Delta r$ yields a gain in the rotor thrust coefficient $C_{TR}$ which goes from 0.62 to 0.74 by ducting the turbine (see Tab. 1), while the augmentation of the tangential-force linear-density $\Delta F_\theta/\Delta r$ gives rise to a growth in the extracted power. This definitively proves that the duct significantly changes the rotor operating conditions and that, contrarily to some common practice, a rotor designed to be employed in the open configuration cannot be also used in the ducted one, if an optimized set-up is sought.

As previously mentioned, another aspect which deserves to be analysed is the differences in the tip flow arising by ducting the rotor. In the ducted case, the flow near the blade tip is driven into the gap by the pressure differences between the pressure and suction side, and by the scraping effect of the duct inner surface. In both cases, the strong acceleration at the pressure side causes the formation of a 3D vortical structure rolling over the rotor tip surface in the proximity of the edge along this side.
Figure 5: Linear density of the axial (left) and tangential (right) forces on a single blade.

Figure 6: Contours of entropy at different meridional planes and static pressure on the blade surface. Left: OWT, right: DWT.

The separation bubble is visualized in Fig.s 6 and 7 through the contours of the entropy and tangential vorticity $\omega_\theta$ reported in several meridional planes uniformly distributed along the blade chord. The figures also show the formation of a secondary contra-rotating vortex beneath the primary blade-aligned vortex tube as well as higher tip losses in the DWT case associated with its bigger entropy values. Moreover, in the OWT case the roll leaves the blade moving towards the trailing edge and sheds near the pressure side. Contrarily, in the DWT, the blade-aligned vortex is larger and lays on the tip surface all over the chord length. On the suction side, the flow coming from the pressure side rolls-up to eventually form the tip vortex which is continuously fed by the flow exiting the tip gap. Also, due to the lower blade load, the tip vortex has a weaker strength in the OWT compared with the DWT. Leaving the blade trailing edge, it sheds in the rotor wake following an helical path and interacts with the duct boundary layer (see Fig. 8). This phenomenon could possibly delay any duct separation onset by re-energizing the low-momentum fluid, as also noticed by Aranake et al. (2015). Downstream of the rotor, the increase in the wake divergence, together with the duct boundary layer interaction, weakens the
vortex strength leading to its breakdown.

CONCLUSIONS

An automated procedure to generate a full matching multi-block hexahedral mesh system has been presented along with the simulation set-up for the analysis of the low-Mach flow past DWTs. The methodology has been successfully validated against available experimental data, and used to analyze the differences in the operating conditions and in the local flow features between an OWT and a DWT. According to a common practice, the latter has been obtained using as rotor the same geometry of the OWT case. The analysis has shown that the DWT swallows an higher mass-flow rate due to the duct bound circulation, which leads to a growth in the extracted power. Specifically, the analysis has revealed that the duct induces a change in the incidence all along the blade span as well as a significant variation in the axial and tangential blade forces. As expected the straightforward ducting of an existing open rotor is not adequate to achieve the maximum attainable performance enhancement since it is unlike that the device will properly operate at optimal flow conditions. Therefore, the design of a DWT
cannot be considered completed with the blunt coupling of its two main components, but the strong mutual interaction between the duct and the rotor must be explicitly taken into account early at the design stage. Finally, some peculiarities of the tip flow have been presented.

References


