Wind Energ. Sci. Discuss.,
https://doi.org/10.5194/wes-2019-62-AC3, 2021

Interactive comment on "Ducted wind turbines in yawed flow: A numerical study" *by* Vinit Dighe et al.

Vinit Dighe et al.

v.v.dighe@tudelft.nl

Received and published: 28 June 2021

The authors appreciate the valuable comments from the reviewers. The manuscript has been modified following the reviewer's comments. Modifications are reported in orange in the revised manuscript. For the sake of completeness, modifications carried following the comments of reviewers 1 and 2 in the first round of peer-review are retained and highlighted in red, modifications carried following the comments of reviewers 3 and 4 in the second round of peer-review are retained and highlighted in blue and modifications carried following the comments of reviewers 5 and 6 in the third round of peer-review are retained and highlighted in green.

1. it is not possible to refer to an experimental work to verify your numerical domain

size. You have to verify this by performing simulations with different domain sizes and report the influence on Ct and Cp. If you are not able to perform such a study then you need to state in the article that the relative small domain might have influenced the results and future work is necessary to quantify it.

To address this remark, a study on domain blockage effects is now included in Appendix A.

2. The Reynolds-numbers that you have mentioned to me cannot be found in the revised version, please add them, because this is important information for those who would like to redo your simulations.

The Reynolds number is now explicitly stated in the article; i.e. Re of 4.5 $\times$ 10^5.

3. Has this been settled? The end of this paragraph seems to suggest that the wind-aligned flow through a ducted turbine is well understood, which this statement appears to contradict. Also: What about blade tip aerodynamics and induction effects? Are the diffusers designed to ensure flow attachment throughout the near wake? Please elaborate.

To address these remarks, Section 1 (Introduction) is modified and the modifications are highlighted in orange.

4. I believe "tau" in the caption should be "Gamma." Also, what you've drawn is the circulation in the wake, which is equal and opposite to the actual bound circulation.

Figure 1 is modified.

5. This equation does not appear to have consistent units–the LHS is dimensionless while the RHS has units of velocity.

Equation 2 is corrected.

6. This is unclear to me. C_{T_D}, which is the force exerted by the duct on the flow, is equal and opposite to the drag force exerted by the wind on the duct. Per the discussion in

the introduction, the increased mass flow should arise from the lift generated by the duct.

To this aim, the related text is re-written in order to avoid confusion.

7. Are you running DES? Otherwise, this is overkill.

The authors appreciate this valuable remark. However, the chosen y+ value is based on the User manual provided by Ansys (see https://www.afs.enea.it/project/neptunius/docs/fluent/html/ug/node410.htm)

8. Is this assumption valid, especially given yawed inflow (e.g., as you've sketched out in Fig 2) at > 20 deg, as discussed later in Section 5?

The AD approach is chosen deliberately for this fundamental research/investigation, so as to study the impact of duct shapes, and not the specific performance of a rotor within a duct. In order to assess the validity of the AD approach, an additional numerical verification exercise of the 2D AD approach is performed in Appendix B, where the results are compared to a full-scale DWT numerical model. The results show that the simplified AD approach is suitable to capture the first order flow physics; for higher order effects, the blade shape resolving models will be well suited.

9. Where does this come from? It appears quite low.

The value is based on the cited author's choice for the experiments, and is used for numerical validation in our study.

10. Is Cp calculated by multiplying the rotor-averaged velocity with the input Ct? Did you run each case with and without the diffuser geometry?

C_P reported here is based on the equation 6, and $\frac{U_{AD}}{U_\infty}$ is the integral of the velocity across the AD surface. Yes, for calculating r (augmentation factor), simulations for AD without duct are performed but the numbers not reported here explicitly for the sake of brevity.

11. Unclear what value this additional exercise adds. It shows that the wake deflection as well as the flow around the far diffuser wall (at +y) differs significantly between the 2D URANS and 3D VLES.

The following exercise was based on the recommendation of a reviewer in order to assess the validity of the AD approach.

12. At what distance downstream of the rotor? Are the 2D and 3D results similarly comparable further downstream?

The downstream distance is now indicated in Figure 6. Unfortunately, no additional measurements were available further downstream to extend our numerical validation.

13. How does the duct thrust continuously decrease for the DonQi design? For example, comparing alpha=20 in Fig 9, there is massive flow separation on both the top and lower parts of the duct; I would expect the thrust to be >= the thrust on the D5 at this point.

The authors highly appreciate for addressing this detail. At α = 20°, where both DonQi® and DonQi D5 configurations are completely stalled, the resultant C_{T_D} is higher for DonQi D5 duct. This is because the impact of stalled flow on the pressure side of the windward airfoil for DonQi D5 is larger since the stagnation pressure acts on the concave duct surface in comparison to the DonQi® duct surface, which is more convex. Hence, the resultant C_{T_D} for DonQi D5 duct is much higher when compared with the DonQi® duct even though the general flow pattern (α = 20°) looks quite similar. A more detailed explanation is added as modifications in the manuscript.

Interactive comment on Wind Energ. Sci. Discuss., https://doi.org/10.5194/wes-2019-62, 2019.