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: 23 April 2021

The authors appreciate the valuable comments from reviewers 5 and 6. The manuscript has been modified following the reviewer's comments. Modifications are reported in green 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, and modifications carried following the comments of reviewers 3 and 4 in the second round of peer-review are retained and highlighted in blue.

Response to reviewer 5

- What is the reason for performing URANS simulations? Couldn't you perform (steady-state) RANS simulations and obtain the averaged quantities directly?
  Large flow separation regions are expected for DWTs in yawed flow depending on the geometry of the duct and the yaw angle. Flow solutions obtained using steady RANS formulation for DWTs with large yaw angles did not converge or even diverge. Using URANS formulation, the goal was to capture the asymptotic behavior (quasi-steady-state) of the flow in order to reach a converged solution. The selected approach also takes the learning from Apsley Leschziner (2000), who investigated the ability of unsteady simulations using the $k-\omega$ SST model to predict separated flows in a duct and compared them to experimental data. The agreement between the experimental and computation results was found to be good, and therefore the approach was chosen for the current investigation.

- I would recommend a more realistic AD model, especially when you investigate the effect of yaw misalignment on the forces. Have you investigated more realistic loading distributions?
  In all the simulations presented in this article, the turbine is represented using a numerical actuator disc (AD) model, a method widely used to model the principal effects of the turbine in a simplified manner. In a recent study by the authors (see: Dighe, Vinit V., Francesco Avallone, and Gerard van Bussel. "Effects of yawed inflow on the aerodynamic and aeroacoustic performance of ducted wind turbines." Journal of Wind Engineering and Industrial Aerodynamics 201 (2020): 104174.), it was shown that the azimuthal variation of axial velocity at the rotor radial plane was relatively low. Incorporating the azimuthal effects using more sophisticated techniques like actuator line or actuator surface methods would definitely enable more accurate calculations of the local induced velocities. However, one should note that the flow physics for a bare turbine and a turbine within a duct is completely different. Incorporating the real rotor geometry/distributed loading for modeling turbine effects during this preliminary investigation would not allow us

to decouple the turbine and the duct effects, thus preventing a proper analysis of the yawed flow for DWTs. To this aim, the simplified AD approach is chosen deliberately for this investigation, so as to study the impact of duct shapes and not the specific performance of a rotor within a duct.

- Do you also assume a constant thrust load and no tangential forces?
  A uniformly (constant) loaded AD model without tangential loading is used to represent the turbine. The main difference between the 2D and the 3D results is that the 3D URANS simulations use the azimuthally averaged streamwise velocity component, while the results from 2D simulations do not account for the gradual variation with the azimuthal angle.

- I was wondering if there exists literature where a ducted wind turbine has been simulated by resolving the geometry of both the duct and wind turbine in the numerical grid. If one had such a model available, then one could verify the loading of the present setup where the wind turbine is modeled as an AD.
  The current article is meant to be a preliminary investigation to study the aerodynamics of DWTs in yawed flow, and particularly on the effect of the duct geometry on the aerodynamic performances. A comparison study with the real DWT model (a subsequent paper published by the authors), in which both the duct and the turbine have been simulated will certainly improve the quality of the numerical verification and validation. Having said that, the numerical verification exercise is added in the article as an appendix.

- You mentioned that you model the AD with an infinitesimal width, which makes sense for a 2D simulation, but not for 3D simulation. Please clarify in the text.
  This has been rectified.

- Have you investigated the effect of this relatively small domain on the results?
  A detailed study to investigate the blockage effects due to the varying domain size has not been investigated in the current research. However, in a previously

published research by the authors (Dighe, V. V., Avallone, F., Igra, O., Bussel, G. V. (2019). Multi-element ducts for ducted wind turbines: a numerical study. Wind Energy Science, 4(3), 439-449.), a similar numerical setup was validated with experiments where the wall interference and blockage correction can be ignored. A good agreement was seen.

- You could split the methodology section into two subsections, describing the 2D and 3D CFD setups, separately
  The section has been revised for clarity.

- What are the Reynolds numbers of all validation cases and how do they compare with the typical Reynolds number of utility-scale urban wind turbines?
  In the context of existing commercial DWT models, the Reynolds number range from 200,000 to 1,000,000 depending on the model geometry. The current numerical study is performed at a fixed Re of 4.5 $\times$ 10^5.

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