Wind Energ. Sci. Discuss.,
https://doi.org/10.5194/wes-2017-54-AC1, 2018

WIND ENERGY SCIENCE DISCUSSIONS

Interactive comment on "Experimental Validation of a Ducted Wind Turbine Design Strategy" *by* Benjamin Kanya and Kenneth D. Visser

Benjamin Kanya and Kenneth D. Visser

visser@clarkson.edu

Received and published: 7 February 2018

Dear Referee #1

Many thanks for your comments and advice.

I have addressed all your questions below on behalf of my co-author as well, however I have not attached a modified manuscript yet. Please let me know if this is sufficient and the answer to a couple questions below.

1. What are the new findings and what are the inventions? The primary finding is that a computational design strategy, involving the evaluation of a numerical solution of the ducted flow field as an input to the design code for the rotor itself, has been experimentally validated. This enables one to design a ducted turbine to a different

size with a much higher degree of confidence in the real world performance. It also enables us to achieve higher Cp values.

2. For the numerical method, what kind of grid type did the author employ? The grid had a boundary layer mesh with a growth rate of 1.1 and the first mesh point was set at yˆ+≈1. The boundary layer thickness was calculated as a function of Re_c for each case and enough inflation layers were used to span the entire boundary layer. A layer of quadrilateral elements covered the actuator disc which was used to model the turbine in our 2D axisymmetric model. There was a refined unstructured triangular grid around duct which was surrounded by a large structured quadrilateral grid covering further the upstream and downstream of the actuator disc. The rest of domain was meshed with unstructured quadrilateral elements (please see the companion article under review for WESC by Bagheri-Sadeghi et al., "Ducted Wind Turbine Optimization and Sensitivity to Rotor Position" for further details)

3. What is the resolution around the turbine? The actuator disc is covered with 200 elements (i.e. for the 2.5m rotor each element in our axisymmetric model covered 6.25 mm)

4. Did the authors pay attentions to the Reynolds number effect and the grid resolution dependence to make clear the flow characteristics around the ducted wind turbine which shows flow separation and reattachment inside the duct, and vortex shedding from the duct? As mentioned above the boundary layer thickness and y+ was calculated for each case to make sure the most accurate results that one can obtain from a RANS CFD solution can be obtained. The k-ω SST turbulence model was utilized, which among the two-equation turbulence model gives better prediction of flow separation.

5. The authors discussed the surface flow feature near the exit of the duct in the 4.2 Flow Field Issues. It strongly suggest that the flow around the ducted turbine both inside and outside of the duct are highly unsteady, unstable and turbulent flows. The

reviewer cannot understand the accuracy of CFD presented in this paper. Although the predictions of power output from the CFD model agreed very well with the experimental results, as you suggested there was a difference between CFD predictions and experimental results. The CFD model showed no flow separation whereas flow separation was observed in the experimental tests. This could be due to simplifications of the CFD model like using a 2D axisymmetric model where the turbine was replaced with an actuator disc, or the limited accuracy of two-equation turbulence models. It could also have resulted from the differences of the manufactured model from the CFD model; from the final geometry of the duct, to the actual physical supporting structures used for the duct, and the influence of the individual rotor blades (as opposed to a uniform disc) which affect the flow field. Lastly, the proximity of the ducted wind turbine to the floor could have aggravated the flow separation.

6. For the wind tunnel experiments, what is the uniformity of approaching wind in the wind tunnel of the University of Waterloo? The uniformity of the wind was not explicitly mapped. The flowfield is generated by a bank of 6 100 hp fans with independent variable speed control. Data was acquired with a single location sonic anemometer. At a given velocity setting the flowfield was sampled with a hand held anemometer and compared to the single point. The average variation of the incoming flow to the turbine varied by a +/- 1-2%.

7. What is the turbulence intensity and its uniformity? The turbulence of the flow filed was sampled over the range of the incoming velocities and was found to vary over the range of 5-10%

8. What is the blockage ratio? The blockage ratio, based on the projected frontal area of the duct and the rotor, for a stationary rotor, was 4.1%. If the rotating rotor is considered to act as a solid blockage, the ratio would be 11.7% No blockage corrections were made.

9. For the ducted wind turbine prototype employed here, the authors should describe

the figure of the curved shape of the duct with the inlet diameter, throat diameter and exit diameter We can include a figure of the detailed geometry or a text description, namely that the airfoil was an Eppler 423 with a chord of 0.6 m with inlet, throat and exit diameters of x, x, and x. Which would the referee feel to be more appropriate?

10. The referee recommends that the author should evaluate the modified Cp which adopts the maximum duct area, i.e., the projected area of the duct as the reference area. For reference, please check a paper of Energies 2010, 3, 634–649; doi:10.3390/en3040634 “A Shrouded Wind Turbine Generating High Output Power with Wind-lens Technology” Yes, the Cp values are scaled by the rotor area. Scaling by the projected are of the duct is, perhaps, a more ‘fair’ evaluation of the data compared to a conventional turbine. In the case here, the exit diameter is 3.3 m and the rotor diameter is 3 m. Hence the Cp values would all be scaled by the ratio 3ˆ2/3.3ˆ2 or a factor of 0.826. Would the referee prefer the data to be shown in a separate plot or on the same plot in Figure 16?

Detailed points 1. p. 4, l 18-20; the author should explain “a” in more detail. And in also Figure 6, what are ωr(1+a’), Vo(1-a). Please describe the definitions of all the symbols in the present paper. A more detailed explanation of a and a’ will be put in the manuscript.

2. Figure 5 b); the scales are written in so small letters The plot scales will be increased in size as will several of the other plots such as Figure 7.

Interactive comment on Wind Energ. Sci. Discuss., https://doi.org/10.5194/wes-2017-54, 2017.