WES2017

Title : Experimental Validation of a Ducted Wind Turbine Design Strategy
Author(s) : B. Kanya and K.D. Visser

Referee's Comments :

This paper describes an experimental and numerical study of the flow around a ducted wind turbine. For the wind tunnel experiment, they used a prototype wind turbine made by Clarkson University with the rotor diameter of 2.5m and duct diameter of 3.3m, it means the flow around the blades is enough in high Reynolds numbers. They used a DNS provided by Fluent not using any turbulence model to analyze the flow around the turbine adopting an actuator disk model. The results obtained are reasonable and the power output coefficient Cp values are comparable with other values reported in the past. The results obtained confirm the findings which were already reported from other research using wind tunnel experiments and CFD. However, the referee wonder what are the new findings and what are the inventions in the present research.

For the overall evaluation, the present paper gives relatively good experimental and numerical results and suitable discussion. However, a couple of discussions seem to be insufficient. The referee described those problems below in the detailed points. It seems to the referee that a further improvement and revise still remain for the present paper. If the authors can give further information, the present paper should be accepted for the WES under this content.

Major points:

For the numerical method, what kind of grid type did the author employ? What is the resolution around the turbine? 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. 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.

For the wind tunnel experiments, what is the uniformity of approaching wind in the wind tunnel of the University of Waterloo? What is the turbulence intensity and its uniformity? What is the

blockage ratio? These are the fundamentals of the specification of a wind tunnel. It seems to the referee that the experimental results from the combination of the present wind tunnel and a wind turbine model, there are a little blockage effect in the results.

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

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"

Furthermore, it is necessary to correct the following points.

Detailed points

1. p. 4, l 18-20; the author should explain "a" in more detail. And in also Figure 6, what are ω r(1+a'), Vo(1-a). Please describe the definitions of all the symbols in the present paper.

2. Figure 5 b); the scales are written in so small letters