Interactive comment on "Numerical Analyses and Optimizations on the Flow in the Nacelle Region of a Wind Turbine" *by* Pascal Weihing et al.

A. Smaili (Referee)

arezki.smaili@g.enp.edu.dz

This paper deals with numerical study of the flow field features within the hub region of wind turbine rotor. The study is focusing on the effect of nacelle geometry upon blade root aerodynamics. For this purpose, the generic version of the Enercon E44 wind turbine nacelle integrating blades with flatback airfoils root has been considered. The finite volume solver of FLOWer code based on compressible Navier-Stokes equations has been used. The manuscript is well presented and organised. However, I have the following remarks.

1. The flow in the nacelle region of a wind turbine has been the subject of several previous studies. Thereby, the authors may improve the literature review, a more serious

bibliographical search and study might be carried out. 2. The mesh study was not presented: the choice of grid type and size was not justified. 3. The use of a compressible Navier-Stokes solver should be justified. It would be desirable to present the contours of the Mach number. 4. The validity of the numerical simulations was not presented. 5. The full-turbulence models are not suitable for describing the flow fields in the hub and nacelle region; because probably in such situation, boundary layer transition may occur, and therefore conclusions drawn might be far from reality. 6. To obtain more relevant conclusions, the simulations should be carried out for other wind speed values, not only for 10 m/s.

Interactive comment on Wind Energ. Sci. Discuss., https://doi.org/10.5194/wes-2018-38, 2018.