عنوان مقاله:

Numerical Investigations on the Fluid Behavior in the Near Wake of an Experimental Wind Turbine Model in the Presence of the Nacelle

محل انتشار:

دوماهنامه مکانیک سیالات کاربردی, دوره 16, شماره 1 (سال: 1402)

تعداد صفحات اصل مقاله: 13

نویسندگان:

A. Bouhelal - *Laboratory of Green and Mechanical Development (LGMD), École Nationale Polytechnique, B.P. ۱۸۲, El-Harrach, Algiers, ۱۶۲۰۰, Algeria*

A. Smaili - *Laboratory of Green and Mechanical Development (LGMD), École Nationale Polytechnique, B.P. ۱۸۲, El-Harrach, Algiers, ۱۶۲۰۰, Algeria*

O. Guerri - *Renewable Energy Development Center (CDER), B.P. ۶۲, Route de l'Observatoire, Bouzaréah, Algiers, Algeria*

c. Masson - *Department of Mechanical Engineering, École de Technologie Supérieure, ۱۱۰۰ Notre-Dame Ouest, H۳C۱K۳, Montréal, Québec, Canada*

خلاصه مقاله:

Accurate predictions of the near wake of horizontal-axis wind turbines are critical in estimating and optimizing the energy production of wind farms. Consequently, accurate aerodynamic models of an isolated wind turbine are required. In this paper, the steady-state flow around an experimental horizontal-axis wind turbine (known as the MEXICO model) is investigated using full-geometry computational fluid dynamics (CFD) simulations. The simulations are performed using Reynolds-Averaged Navier-Stokes (RANS) equations in combination with the transitional k-kl-w turbulence model. The multiple reference frame (MRF) approach is used to allow the rotation of the blades. The impacts of the nacelle and blade rotation on the induction region and near wake are highlighted. Simulation cases under attached and detached flow conditions with and without the nacelle were compared to the detailed particle image velocimetry (PIV) measurements. The axial and radial flow behaviors at the induction region have been analyzed in detail. This study attempts to highlight the nacelle effects on the near wake flow and on numerical prediction accuracy under various conditions, as well as the possible reasons for these effects. According to simulation results, the rotation of blades dominates the near wake region, and including the nacelle geometry can improve both axial and radial flow prediction accuracy by up to ۱۵% at high wind speeds. At low wind speeds, the nacelle effects can be ignored. The presence of the nacelle has also been shown to increase flow separation at the trailing edges of the blade airfoils, increasing both root and tip vorticities. Finally, small nacelle diameters are recommended to reduce flow separation on the blades and increase the average velocity downstream of the rotor, thereby optimizing wind farm output power.

کلمات کلیدی:

Wind turbine, Aerodynamics, Near wake, Nacelle-blade interaction, CFD, Mexico