Fluid-structure interaction computations for geometrically resolved rotor simulations using CFD

This paper presents a newly developed high-fidelity fluid–structure interaction simulation tool for geometrically resolved rotor simulations of wind turbines. The tool consists of a partitioned coupling between the structural part of the aero-elastic solver HAWC2 and the finite volume computational fluid dynamics (CFD) solver EllipSys3D. The paper shows that the implemented loose coupling scheme, despite a non-conservative force transfer, maintains a sufficient numerical stability and a second-order time accuracy. The use of a strong coupling is found to be redundant. In a first test case, the newly developed coupling between HAWC2 and EllipSys3D (HAWC2CFD) is utilized to compute the aero-elastic response of the NREL 5-MW reference wind turbine (RWT) under normal operational conditions. A comparison with the low-fidelity but state-of-the-art aero-elastic solver HAWC2 reveals a very good agreement between the two approaches. In a second test case, the response of the NREL 5-MW RWT is computed during a yawed and thus asymmetric inflow. The continuous good agreement confirms the qualities of HAWC2CFD but also illustrates the strengths of a computationally cheaper blade element momentum theory (BEM) based solver, as long as the solver is applied within the boundaries of the employed engineering models. Two further test cases encompass flow situations, which are expected to exceed the limits of the BEM model. However, the simulation of the NREL 5-MW RWT during an emergency shut down situation still shows good agreements in the predicted structural responses of HAWC2 and HAWC2CFD since the differences in the computed force signals only persist for an insignificantly short time span. The considerable new capabilities of HAWC2CFD are finally demonstrated by simulating vortex-induced vibrations on the DTU 10-MW wind turbine blade in standstill. Copyright © 2016 John Wiley & Sons, Ltd.