Wind and Turbulence; Aero-servo-elasticity, loads and structures.

Comparison between a Chimera technique and sliding interfaces for fluid-structure interaction simulations of wind turbines

G. Santo[a], M. Peeters[b], W. Van Paepegem[b] and J. Degroote[a]

Fluid-structure interaction (FSI) simulations always involve a deforming fluid domain. Additionally, turbomachines such as wind turbines have rotating components. Consequently, the deformation of the fluid domain in FSI simulations of wind turbines consists of a rigid body rotation combined with aeroelastic deflection of the blades. This work presents a comparison between two techniques to handle the deforming fluid domain in FSI simulations of wind turbines, namely a Chimera technique and sliding interfaces. In the Chimera simulations, a rotating body-fitted grid is attached to the blades and it overlaps with a stationary background grid. An alternative approach is the adoption of sliding interfaces to separate the stationary domain from a rotating cylindrical one which is body-fitted to the rotor (1). Both techniques rely on deformation of the body-fitted grid to account for the aeroelastic deflection.

The test case is a horizontal axis wind turbine with 3 blades, which will be considered flexible, and a diameter of 100m. The transient fluid-structure interaction is simulated by an in-house code which couples two solvers, one for the computational fluid dynamics (CFD) and one for the computational structure mechanics (CSM). Strong coupling is applied as the force and displacement equilibriums are always enforced on the fluid-structure interface. The CSM model accurately reproduces the details of the composite material and it includes the outer layups, the inner shear webs and the adhesive joints. On the CFD side, the atmospheric boundary layer (ABL) is modelled by the adoption of the $\kappa-\varepsilon$ turbulence model in combination with modified wall functions on the bottom of the domain. The ABL velocity profile is visible in the velocity magnitude contours acquired during the wind turbine rotation as displayed in Figure 1.

The simulations are carried out at the best efficiency point (BEP) of the wind turbine. The power output and the deflection of the blades are monitored throughout the rotation. The results and the computational effort of both techniques are analysed.

[a] Department of Flow, Heat and Combustion Mechanics, Ghent University, Sint-Pietersnieuwstraat 41 – 9000 Ghent, Belgium
[b] Department of Materials Science and Engineering, Ghent University, Technologiepark-Zwijnaarde 903 - 9052 Zwijnaarde, Belgium

[1] Hsu et Bazilevs, *Computational Mechanics* **50,** 821 (2012).

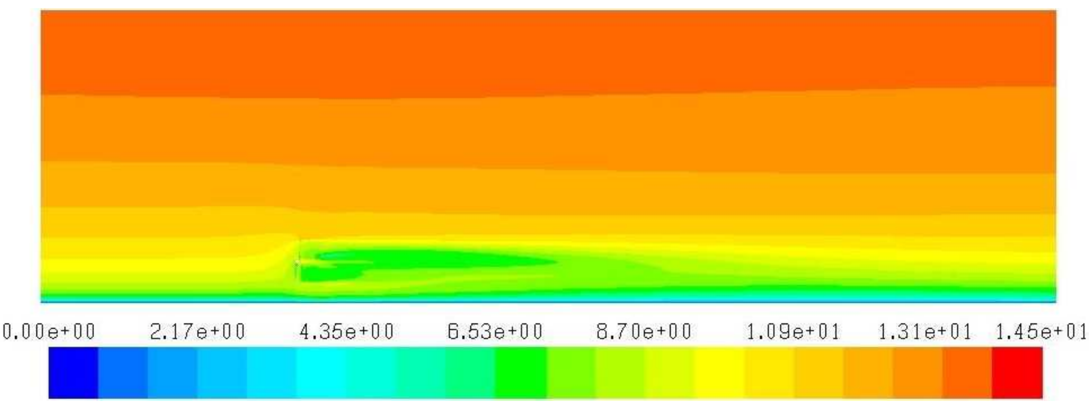

Figure 1: Contours of velocity magnitude [m/s] in a vertical plane containing the rotor's axis of rotation.