Improving 6-DoF motion capabilities of interFoam for floating offshore wind applications

Johan Roenby[a,b], Hamid Sarlak Chivaee[c] & Henrik Bredmose[c]

a STROMNING, Luftmarinegade 62, 1432 København K, DK, johan@stromning.com, Ph: +4529931923
b Department of Mathematical Sciences, Aalborg University, DK
c DTU Wind Energy, Technical University of Denmark, DK

One of the key challenges in the design of reliable and cost-effective floating offshore wind turbines is the assessment of the floater rigid body motion response to extreme waves encountered by the floater during its lifetime. Here CFD in general and OpenFOAM in particular can be a valuable tool for assessing feasibility of a proposed floater concept in the pre-design phase. The OpenFOAM software components required for such studies are all available as open source: The interFoam solver (previously interDyMFoam) in conjunction with the dynamic mesh and sixDoFRigidBodyMotion libraries, various wave generation/absorption and mooring line libraries. However, several studies report problems limiting the reliability, robustness and applicability of the currently implemented 6-DoF rigid body-CFD coupling approach for floater studies[1]–[3]. In particular, the solver suffers under certain circumstances from stability issues related to the added mass effect. This can partially be compensated by increasing the number of outer correctors and/or applying under-relaxation on the floater acceleration. However, outer correctors are computationally expensive and even dynamic under-relaxation does not (to our knowledge) guarantee stable coupling. As part of the recently initiated FloatStep project funded by Innovation Fund Denmark, we are currently working on improving the 6-DoF-CFD coupling methodology of interFoam. The aim is to develop a novel coupling algorithm, which is stabilised by consistently taking into account the instantaneous fluid momentum/energy increase associated with acceleration of the floating rigid body. In the presentation, we will show preliminary results for our new coupling mechanism and discuss further required development for turning interFoam into a valuable design tool for offshore floater concepts.

[1] B. Devolder, P. Schmitt, P. Rauwoens, B. Elsaesser, and P. Troch, “A Review of the Implicit Motion Solver Algorithm in OpenFOAM® to Simulate a Heaving Buoy,” presented at the 18th Numerical Towing Tank Symposium (NuTTS18), Cortona, Italy, 2015, p. 7.
[2] A. J. Dunbar, B. A. Craven, and E. G. Paterson, “Development and validation of a tightly coupled CFD/6-DOF solver for simulating floating offshore wind turbine platforms,” *Ocean Eng.*, vol. 110, pp. 98–105, Dec. 2015.
[3] N. G. Jacobsen, D. R. Fuhrman, and J. Fredsøe, “A wave generation toolbox for the open-source CFD library: OpenFoam®,” *Int. J. Numer. Methods Fluids*.