6th European Conference on Computational Mechanics (ECCM 6)
7th European Conference on Computational Fluid Dynamics (ECFD 7)
11 – 15 June 2018, Glasgow, UK

Navier-Stokes CFD Analysis of Tidal Stream Turbine Current and Wave Loads

Federico Attene[1], M. Sergio Campobasso[2]

[1] Lancaster University, LA1 4YW United Kingdom,
f.attene@lancaster.ac.uk

[2] Lancaster University, LA1 4YW United Kingdom,
m.s.campobasso@lancaster.ac.uk,

Key Words: Navier-Stokes CFD, Tidal Stream Turbine Hydrodynamics, Wave Loads.

This study focuses on assessing the predictive capabilities of Navier-Stokes (NS) Computational Fluid Dynamics (CFD) of steady current induced loads and unsteady surface wave induced loads acting on the rotors of horizontal axis tidal stream turbines (TSTs). The research builds on the preliminary ANSYS FLUENT CFD analyses of [1], a study which considered a TCT towing tank experiment [2], in which steady flow conditions were obtained by towing a model TCT rotor with a track-sliding main carriage, and wave-like inflow conditions were achieved by using a secondary carriage between the main carriage and the rotor to impose an additional harmonic axial velocity component of the rotor. The steady and time-dependent CFD analyses of [1] modelled a simple 120 degree-grid sector, neglecting tank blockage and free surface effects, and also not including the rotor hub geometry. The agreement between measured and computed steady torque, in-plane and out-of-plane blade root bending moments was found to be fair at 73 RPM for low and optimal tip speed ratios (TSRs), but notably worse at 96 RPM. The difference in the level of agreement of measured and computed out-of-plane blade root bending moment was instead comparable.

It was assumed that the aforementioned discrepancies between computed results and measured data could be due to the lack of tank blockage modelling, the dependence of the tank blockage on the TSR, and partly to the omission of the rotor hub geometry in the CFD model. The main objective of this work is to validate this assumption by performing new ANSYS FLUENT CFD analyses modelling the tank geometry, thus including tank blockage effects, and also including the hub geometry in all CFD models. In addition to investigate the dependence of tank blockage effects on the steady performance of the turbine, the study will also investigate numerically the impact of tank blockage on the unsteady loads for the case in which the oncoming water speed varies periodically to approximate the effects of surface gravity waves. To reduce computational costs, both the steady and time-dependent simulations use the frozen-rotor set-up, whereby the flow past the rotor is solved in the rotating frame of reference, which is motionless in both steady and time-dependent simulations. The flow unsteadiness of the time-dependent simulations representative of wave conditions is due merely to the user given velocity perturbation at the inflow boundary. The presented work also considers the impact on modelling laminar-to-turbulent transition on the quality of the steady and time-dependent CFD predictions, measured by comparison with available experimental data.

REFERENCES

[1] G. Amato, S. Doyle, S. Petley, S. Campobasso, M.S. Milne and I.A. Aggidis, Navier-Stokes CFD Analysis of a Tidal Turbine Rotor in Steady and Planar Oscillatory Flow. *European Wave and Tidal Energy Conference*, 27th-2nd September, Cork, Ireland, 2017.

[2] I.A. Milne A. Day, A. Sharma, R.N. Flay and R.G.J, Blade loads on tidal turbines in planar oscillatory flow. *Ocean Eng*. Vol. **60,** pp.163-174, 2013.