CFD analysis of a horizontal-axis turbine in the framework of a blind-test tidal benchmarking project

Danilo Calcagni, Francesco Salvatore, Roberto Muscari
Intitute of Marine Engineering, CNR-INM, Rome, Italy

ABSTRACT

The present work deals with the assessment of variable fidelity simulation models to analyze the complex phenomenology of a turbine operating in a tidal stream. In the framework of an Unsteady Loading Tidal Turbine Benchmarking Study by Supergen ORE HUB [Tucker Harvey et al.(2021)], performances of an Horizontal Axis Tidal Turbine (HATT) are extensively analysed. A general purpose finite volume solver based on the solution of the unsteady Navier-Stokes equations for multi-block structured grids is considered [Dubbioso et al.(2019), Gregori et al.(2020)]. The turbine has been simulated over a wide range of tip-speed ratios, by means of unsteady RANS simulations, in the frame of reference fixed to the rotating turbine, in uniform onset flow and calm water conditions. Since the work is also focused on the assessment of design-oriented hydrodynamics models, cross-validation studies between URANS and BIEM (Boundary Integral Equation Model) results are presented. The wake-field by URANS is presented and analysed with the aim to validate results by lower fidelity models. The wake-field has been processed and geometrical parameters of the trailing wake geometry were derived, allowing, in principle, the improvement of simplified wake models as those adopted by the BIEM.

KEY WORDS: Tidal energy; Horizontal-axis turbine; URANS; Blind test; BIEM; Wake geometry.

INTRODUCTION

The exploitation of the kinetic energy of marine currents has been making great progress in recent years, and represents one of the most promising technology in the marine renewables sector, partly driven by the developers’ experience of the accelerating capability by the application of Computational hydrodynamics models in the turbine design process. The variety of computational models depends mainly on the assumptions, that influence the nominal accuracy of the model but also its computational burden. Depending on the accuracy level, models are considered as high-fidelity and low-fidelity models. The terminology 'Computational Fluid Dynamics' (CFD) usually denotes high-fidelity models, based on the solution of the Navier-Stokes equations, such as Direct Numerical Simulation (DNS, [Nakhchi et al.(2022)]), Large Eddy Simulation (LES) or Reynolds Averaged Navier Stokes (RANS) ([Afgan et al.(2013)]), with different levels of accuracy (time and spatial scales resolved) and, according to it, of different computational burden. These CFD solvers are characterized by the need to provide a computational mesh of the fluid region of interest. By neglecting viscosity effects, the governing equations are simplified and lower-fidelity potential flow models are formulated. A classical computational modelling approach is based on the Boundary Integral Equation Method (BIEM), with the problem solved at the solid boundaries of the fluid region, [Baltazar (2015)]. When the flow is treated according to the momentum theory, loads exerted by lifting bodies’ sections are evaluated on the basis of the local flow-field. Numerical methods are indicated as 'Blade Element Momentum Theory' (BEMT or BIEM, [Allsop et al.(2016)]) and are based on the a-priori knowledge of the aerodynamic characteristics of the sections. The assessment of simulation models to analyze the complex phenomenology of a turbine operating in a tidal stream is a critical point and the subject of ongoing research projects, following a natural priority cascade from high- to low-fidelity models. A common way to produce general reliable results for developers and to provide useful indications for testers is represented by the blind tests. In particular, an experimental campaign is designed around a test case with specific aims that can be oriented both to the quantities of interest (e.g. the global performances of the tested device and/or the resulting flow-field) and to the operating conditions (e.g. evaluate the single device effect or the multi-body interaction or the environmental effect). In the past, several blind test workshops have been prompted and, in the following, a non exhaustive list is provided. In the marine field, a workshop on marine propeller test case, operating in uniform as well as in wake-field conditions, cavitating and not, has been organised to compare results by a number of numerical models, including CFD solvers as Reynolds Averaged Navier-Stokes (RANS), Large-Eddy Simulation (LES), and inviscid-flow methods, [Salvatore et al.(2008)]. The SIMMAN workshop series, [AA.VV. (2008)], have been designed to verify and validate ship manoeuvring simulation methods through CFD solvers. In the renewable energy sector, a workshop was dedicated to a wind turbine wake modelling, tested in isolated and in array configuration, offering a comparative study among RANS, hybrid Actuator Disc/LES, Blade Element Method (BEM) models [Krogstad and Saetran (2020)].

The present work deals with the horizontal-axis turbine test case operating in the marine environment. In this context, a comprehensive tidal turbine bench-marking, the Unsteady Loading Tidal Turbine Benchmarking Study, has been recently proposed by Supergen ORE HUB [Tucker Har-