Blade-Explicit Fluid-Structure Interaction of a Ducted High-Solidity Tidal Turbine

Mitchell G. Borg, Qing Xiao*, Atilla Incecik
Department of Naval Architecture, Ocean, and Marine Engineering, University of Strathclyde, UK

Steven Allsop, Christophe Peyrard
EDF R&D (Electricité de France Research & Development), Chatou, France

Summary: This work elaborates a computational fluid dynamic (CFD) model utilised in the investigation of the structural performance concerning a ducted high-solidity tidal turbine in aligned and yawed inlet flows. Analysing the hydrodynamic performance at aligned flows portrayed the distinctive power curve at which energy is transferred via the fluid-structure interaction. At distinct bearing angles with the axis of the turbine, variations in the blade-interaction due to the presence of the duct was acknowledged within a limited angular range at distinct tip-speed ratio values. As a result of the hydrodynamic analysis, a structural investigation of the blades was discretely evaluated in an effort to acknowledge fluid-structure phenomena.

Introduction

Effectively harnessing the power of the ocean for sustainable energy generation is an incredible feat. In an effort to increase the capacity of energy-generating systems, design altercations have been in constant assessment and development, with a plethora attaining implementation within the global market. On the forefront of the pertinent research in achieving this endeavour is the increase of mass flow through the turbine, together with the alignment of the flow to facilitate further turbine installations. As a result of the development attained, ducts have been installed along the perimeter of rotors to attain an increase in power exchange [1, 2]. The numerical analysis elaborated in this study describes a real-scale CFD model developed to assess the hydrodynamic performance of a high-solidity open-centre tidal turbine within a bidirectional duct in flows at aligned and angular bearings. In continuation of analysing the fluid-structure effects, the structural mechanics of blades was analysed by employment of Finite Element Analysis (FEA), in effort of comprehending the physics induced by the duct.

Methods

In continuation to Borg et al. [3, 4], the CFD models were solved by means of ANSYS Fluent 18.0, where the physical models were designed to consist of a domain layout imposed with relevant boundary conditions. The CFD solver was utilised to compute the Reynolds-averaged conservation equations as time-averaged representations of the continuity and momentum equations which govern the three-dimensional, unsteady, incompressible fluid flow. The domain surrounding the turbine was segregated from the global domain to induce a moving mesh model with rotation at the turbine. Closure of the Unsteady Reynolds-Averaged Navier-Stokes (URANS) equation was modelled by means of the RSM turbulence model due to its superiority in analysing anisotropic flows.

In an effort to attain a validated CFD model for tidal turbine applications, simulations were established to replicate the experimentation undertaken by Mycek et al. Identical blade, nacelle, and mast geometry were utilised within the model domain; the parameters of the turbine and fluid flow were also instated from the literature. Upon validation, the model settings were implemented for the analysis of a ducted eight-bladed tidal turbine, similar to the design of the OpenHydro PS2 device, illustrated in Figure 1. The turbine geometry was implemented within the model domain. Similar to the validation, the physical model was designed to consist of identical domain layout and boundary conditions as the three-bladed horizontal-axis tidal turbine (HATT). The parameters of the turbine and fluid flow were instated from real-world data, provided by EDF R&D. Subsequent to the hydrodynamic analysis, the pressure distribution along the blade surfaces in variation with elapsed time, was exported from the CFD solver to the FEM solver. The structural parameters of the rotor blades were derived by means of literature with specific regards to the related layout and domain within which the turbine operates; utilising a transient analysis, the outcomes were deduced in an effort to acknowledge commissionable properties for operation.

---

* Corresponding author.
Email address: qing.xiao@strath.ac.uk

Results

Consequent to the methodology, the resultant outcomes, in terms of power coefficient, thrust coefficient, and velocity profiles in the wake, were established for all simulation cases. Primarily, the three-bladed HATT CFD model outcomes were compared to literature. In comparison to the distinct curves, a similarity index of over 95% was acknowledged at the power coefficient plateau region, together with all CFD TSR data points falling within $2\sigma_{CP}$ with experimentation TSR data points, displaying good comparison, as illustrated in Figure 2.

Once the modelling techniques were implemented for the ducted turbine, unique outcomes were displayed. Notably, the TSR curve is relatively short spanning, with a TSR range of 1 – 2.5, which is characteristic of a high-solidity turbine. In this region, the peak power coefficient of 0.34 is achieved along with a decrease of 0.1 from its nominal TSR. In continuation, the implementation of the mechanical properties of the blades permitted the structural analysis to provide outcomes representing the deflections, principal stresses, and frequency in relation to fatigue investigations.

Conclusions

This study put forward the concept of a numerical analysis of a ducted high-solidity open-centre tidal turbine in an effort to establish its hydrodynamic and structural properties in aligned and yawed flow. By means of CFD validation, the power output, thrust resistance, and maxima stresses within the ducted turbine were recognised to depict outcomes for a characteristic representation of the arrangement in real-ocean conditions.

Acknowledgements:

The research work disclosed in this publication is partially funded by the Endeavour Scholarship Scheme (Malta). Scholarships are part-financed by the European Union - European Social Fund (ESF) - Operational Programme II -- Cohesion Policy 2014-2020: "*Investing in human capital to create more opportunities and promote the well-being of society*".

Results were obtained using ARCHIE-WeSt High Performance Computer (www.archie-west.ac.uk).

References:

[1] C. Belloni, "Hydrodynamics of Ducted and Open-Centre Tidal Turbines," University of Oxford, Oxford, United Kingdom, 2013.

[2] S. Allsop, "Hydrodynamic modelling for structural analysis of tidal stream turbine blades," University of Exeter, Exeter, United Kingdom, 2018.

[3] Borg, M. G., Xiao, Q., Incecik, A., Allsop, S., and Peyrard, C., 2018. "Numerical Analysis of a Ducted High-Solidity Tidal Turbine". In OCEANS'18 MTS/IEEE Kobe, IEEE.

[4] Borg, M. G., Xiao, Q., Incecik, A., Allsop, S., and Peyrard, C., 2018. "An Actuator Disc Analysis of a Ducted High-Solidity Tidal Turbine", to be presented at ASME 2019 38th International Conference on Ocean, Offshore and Arctic Engineering OMAE 2019-96014

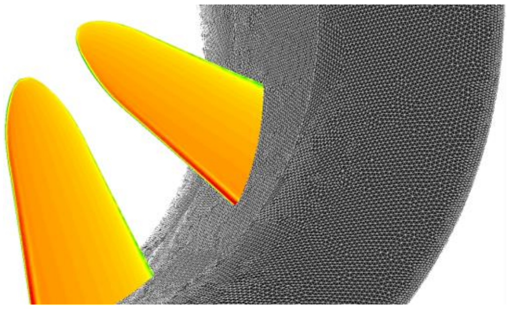

Figure 1 – Ducted High-Solidity Open-Centered Turbine in Representation of Pressure Contours

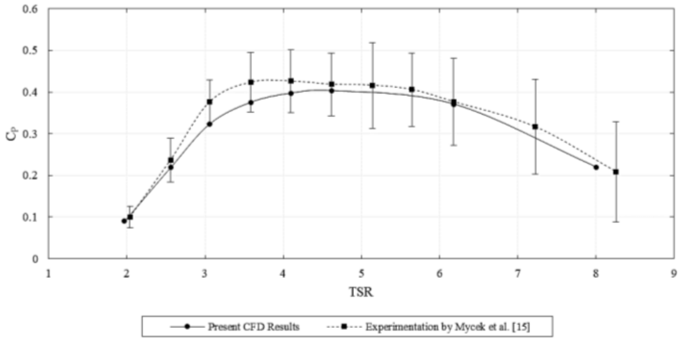

Figure 2 – Comparison of CFD and Experimentation Power Coefficient Results for Three-Bladed HATT [3]