Renewable Energy 78 (2015) 609–620

Contents lists available at ScienceDirect

Renewable Energy

journal homepage: www.elsevier.com/locate/renene

The effect of tidal flow directionality on tidal turbine performance characteristics

C. Frost*, C.E. Morris, A. Mason-Jones, D.M. O'Doherty, T. O'Doherty

School of Engineering, Cardiff University, Queen's Buildings, The Parade, Cardiff, CF24 3AA, United Kingdom

ARTICLE INFO

Article history:
Received 24 February 2014
Accepted 22 January 2015
Available online 11 February 2015

Keywords:
Turbine characteristics
Tide directionality
CFD
Blade rotation

ABSTRACT

With many Tidal Energy Conversion (TEC) devices at full scale prototype stage there are two distinct design groups for Horizontal Axis Tidal Turbines (HATTs). Devices with a yaw mechanism allowing the turbine to always face into the flow, and devices with blades that can rotate through 180° to harness a strongly bi-directional flow. As marine turbine technology verges on the realm of economic viability this paper reveals the performance of Cardiff University's concept tidal turbine with its support structure either upstream or downstream and with various proximities between the rotating plane of the turbine and its support stanchion. Through the use of validated Computational Fluid Dynamics (CFD) modelling this work shows the optimal proximity between rotor plane and stanchion as well as establishing, in the given context, the use of a yaw mechanism to be superior to a bi-directional system from a performance perspective.

1. Introduction

Tidal stream technology is one of the most recent forms of renewable energy to be developed as it offers predictable and regular electrical generation at higher power densities than other renewable energy resources [1]. However tidal stream turbines (TSTs) are best utilised to exploit areas where the flow is constrained by land and seabed topography such as islands and straits [2], and the current is accelerated to greater than 2.5 m/s to make their deployment cost effective [3]. The Department of Trade and Industry report on the economic viability of a simple tidal stream energy capture device [4], and UK resource estimates from Black and Veatch [3] suggest that typical water depths at the suitable sites around the UK range between 25 and 40 m and that consequently the corresponding recommended rotor diameter is between 10 m and 20 m. Although the rotor should ideally be placed as high as possible in the water column, to maximise the available power, it is not always practical to locate them there due to shipping constraints. For, example, the rotational axis of any horizontal TST placed in the Severn Estuary would need to be located 10 m above the seabed in a 35 m depth [5].

Since the rotors are never placed in isolation, but are typically housed on a support structure, it is important not only to characterise the performance of the rotor, but also to fully understand the interaction of the support structure on the flow characteristics. However, little work has been published on the direct effect of a support structure on the performance of a TST, especially when the support structure is upstream of the blades, as could be the case for turbines operating in dual-direction tidal flows. Prior work, carried out by Mason-Jones et al., [6] initially investigated the effect of the stanchion geometry for a horizontal TST, positioned 2 hub diameters or 3.6 m downstream of the rotor, on the characteristic performance of a TST. Different cross-sectional geometries were used to study axial thrust loading on the stanchion. Five different cross-sectional geometries were tested with an additional model without any supporting structure to give baseline values. The effects of these different cross-sectional geometries on the axial thrust are shown in Fig. 1, with a uniform velocity of 3.086 m/s. Although a stanchion with an elliptical or hydrofoil cross-section could be argued to provide the optimal stanchion design, a circular stanchion would be easier to manufacture and as such the circular stanchion was proposed as the stanchion design, based on a compromise between the various factors. As such the circular stanchion geometry will be used in this study.

Experience and knowledge gained from the wind industry has shown that the supporting structure always interferes with the fluid flow around the turbine blades due to the so-called tower dam effect as the flow is retarded in front of the supporting structure [7]. When the rotor is upstream of the supporting structure, the effects are minimal. However, when the blades are downstream, or in the

---

* Corresponding author. Tel.: +44 (0)29 2087 5905; fax: +44 (0)29 2087 4597.
E-mail address: frostc1@cf.ac.uk (C. Frost).

http://dx.doi.org/10.1016/j.renene.2015.01.053

Nomenclature

D	turbine diameter (m)
r	radial distance (m)
R	turbine radius (m)
λ	tip speed ratio
CFD	computation fluid dynamics
TST	tidal stream turbine
L_n	clearance distance between turbine and support structure (m)
ω	rotational velocity (rad/s)
V	free stream velocity (m/s)
R	turbine radius (m)
ρ	density (Kg/m^3)
P	power (w)
F	force (N)
T	torque (Nm)

shadow of the supporting structure, the blades must pass through a sheltered area and this causes significant problems. One possible method to reduce the impact of this would be to increase the clearance distance between the turbine and support structure as this is expected to lessen the impact of the blockage on the turbine. There will be an economic and physical limit to the size of the clearance distance. Only by obtaining the characteristics of the turbine and flow for various clearance distances, however, can this benefit be defined. An alternative method of avoiding this situation would be to always face the blades into the free-stream velocity. This is reasonable within the wind industry where a yaw drive is simpler to incorporate and maintain. In the tidal stream environment, however, the flood and ebb tides would need to be considered to maximise the power generation. Although it may be technically feasible, to rotate the turbine to always face the oncoming flow, the added complexity and the harsh operating environment, let alone the increased capital costs and likely maintenance requirements, mean that the benefits from this option must be substantial. Hence it is important to fully understand the stanchion shadow effects on the turbine when upstream and downstream of a tidal stream turbine. As such a study using CFD has been undertaken to investigate the importance of turbine-stanchion proximity and arrangement on the performance of a 10 m diameter, 3 bladed horizontal axis tidal stream turbine which utilised a Wortmann FX 63-137 profile, with a 33° twist from the blade root to the tip. In particular, the study was conducted to assess how the tidal interaction with the supporting structure affected the performance characteristics of the turbine (i.e. power, torque, thrust and axial bending moments) for various clearance distance, as well as when the turbine is arranged either upstream or downstream of the support structure.

2. CFD

2.1. Model specifications and parameters

In order to understand the fundamental interaction between a blade passing in front of or behind a stanchion, a 3D model was created to establish the flow characteristics. The 3D modelling first determined the effects of stanchion clearance distance on the operational performance characteristics of the TST using a series of steady-state CFD models. Secondly, a transient model was then studied to determine the time dependent variables for the turbine.

The steady-state and transient models consisted of a control volume as defined in Table 1. With the 10 m diameter turbine located 100 m downstream of the inlet and its axis of rotation 25 m below the surface. The turbine is supported by a 2.4 m diameter stanchion which penetrates the complete water column. The axial clearance distance between the back edge of the blade and the front of the stanchion varied between L_1, L_2 or L_3 which are 1.8 m, 2.8 m & 3.8 m respectively either upstream or downstream. For the remainder of this paper these terms will be used to describe the various cases, for example 'L_1 Upstream' would refer to the 1.8 m separation distance between the support structure and the upstream turbine. The L_1 value corresponds to the hub diameter of Cardiff University's turbine. Distances beyond L_3 were viewed as unviable because of the large bending moments that would be incurred, and are not presented in this paper.

The turbine was enclosed by a cylindrical domain which was axially aligned with it. The cylinder and turbine were subtracted from the outer rectangular control volume to form a rotating frame of reference (RFR) with a general grid interface (GGI) connection between the sea and turbine volumes. Conservation of all terms are preserved across the connection interface, however the interface allows centrifugal and Coriolis momentum terms to be computed and solves a rotating frame total energy equation as described in ANSYS guide [8]. This arises from pre-defining the angular velocity of the rotating domain which corresponds with each tip speed ratio (TSR). The steady state model is time independent and provides a

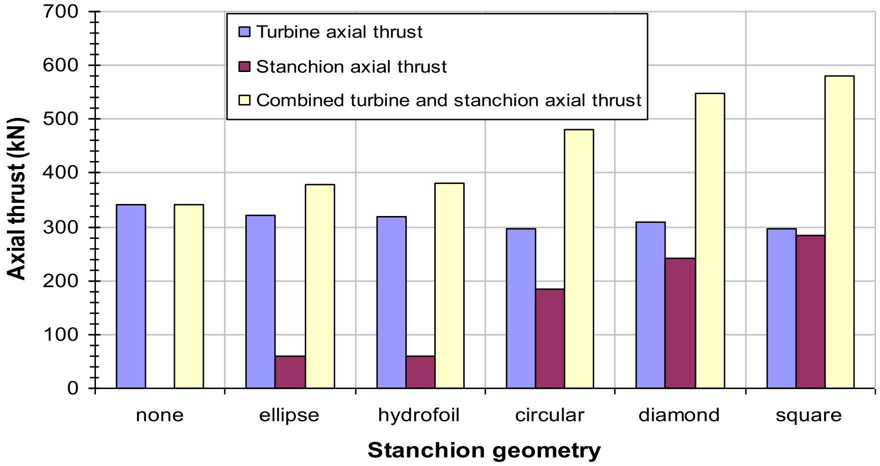

Fig. 1. Effect of stanchion geometry on turbine power extraction and axial thrust.