Modelling Pressure Changes in the Vicinity of Tidal Turbines to Assess Fish Survival Rate During Turbine Passage

Enayatollah Zangiabadi*, Ian Masters, Alison J. Williams, T.N. Croft
College of Engineering, Swansea University, SA2 8PP, UK

Allan Mason Jones
School of Engineering, Cardiff University, CF24 3AA, UK

Summary: Estuaries and channels, which are potential deployment sites for Tidal Stream Turbines (TST), are home to many marine mammals and fish species. One issue that has to be considered before the installation of TSTs is the interaction of these devices with the resident fauna. The pressure change, cavitation and turbulence are among the main factors that endanger the animal during passage through the turbine [1]. In this article, by using BEM-CFD and blade resolved CFD methods, the survival rate for fish is investigated for single and multiple tidal turbine arrays. The results show that if a fish passes through the swept area of the turbine, the majority of the species are able to survive the incident.

Introduction

Apart from the risk of being hit by the blades of the turbine, the other major threat for the fish is the rapid pressure change as it passes through the disk that describes swept area of the turbine. There is a pressure increase in front of the turbine which is followed by a sudden pressure drop immediately behind the turbine. Then the pressure gradually increases again to ambient pressure. For most fish, the main cause of mortality is the damage caused to the internal organs, in particular to the swimbladder. The drastic change in the volume of the swimbladder due to the almost instantaneous pressure change is the main reason for fish mortality[2]. Less significant are the cavitation and turbulence imposed into the water by the TST.

Simulations have been performed for a single device and triangular array configuration using the steady state k-ε turbulence method. The results are then compared with experimental data which were available mainly for the survival of fish going through hydroelectric dams.

Methods

Two different approaches have been used to simulate the fluid flow around a tidal stream turbine. The first approach, which has a shorter run time, uses the Blade Element Momentum Theory coupled with Computational Fluid Dynamics (BEM-CFD) method. In this method the turbine effects are time averaged over a long period of time so that the influence of the blades varies according to radial position based on the property of the blades (chord, twist, etc.) at the given location. [3]. The second takes into account the full turbine Blade Resolved Geometry (BRG) and uses a transient CFD model, resulting in significantly longer run times.

Pressure values are measured along stream traces which are the likely swimming path for the fish. Based on the maximum and minimum values of the pressure, the high risk and low risk location along the swimming path relative to the turbine were identified.

In order to have a direct comparison between the two models, a standard finite volume approach with a k-ε turbulence model has been employed to conduct both simulations. The BRG model, including the Finite Volume model construction and set up, was provided by the Marine Energy Research Group of Cardiff University. It has a 10 m diameter turbine in a rectangular domain with the dimension of $506 \times 50 \times 50$ meters in which the turbine is located 104 meters from the inlet. The inlet boundary conditions are defined as a uniform flow in which the flow speed is 3.086 m/s and the bed has a non-slip wall condition. To run the simulation, the rotational speed of the blade is set to 2.25 rad/s which is the optimum rotational speed for the rotor in normal operation. The zone that represents the rotation of the rotor, has a 17 m diameter and 6 m width. The BEM-CFD model for the single array turbine has exactly the same dimensions and boundary conditions.

For the triangular array the computational domain consists of a rectangular channel with 700 m length, 200 m width and 30 m depth. The turbines are positioned so that the centres are at the midpoint of the channel depth, with the first row of turbines located 300 m from the inlet. The configuration has two turbines in the first row which are 15 m apart. The second row turbine is located 75 m downstream at a lateral position that is midway between the first row turbines to form a triangular shape TST array.

* Corresponding author:
Email address: 502830@swansea.ac.uk

Results

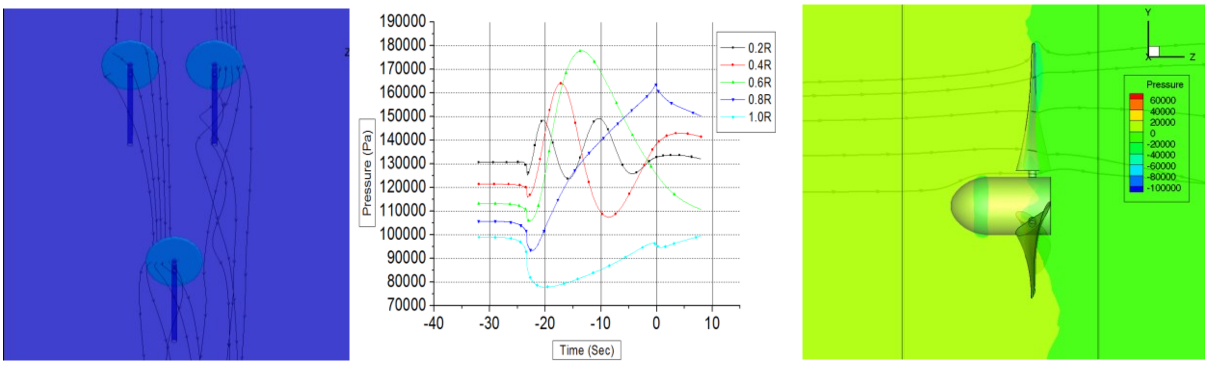

Fig. 1. Stream lines along the length of the channel and through the turbines (left); Pressure Change along the Vertical Section Perpendicular to the Hub of a Tidal Turbine for the First and Second Row of Turbines in the Triangular Configuration for a Tidal Array (middle); Stream lines passing adjacent to the blades of a 10m diameter rotor (right)

The computational results are compared with some existing data:

Radius	0.2R	0.4R	0.6R	0.8R	1.0R
Pressure Change (kPa)	4.4	4.1	4.4	4.7	3.5

Table 1 Pressure Change along the Length of the Blade for a 5m Radius Turbine in the First Row of a Triangular Configuration Tidal Array

Species	Death	Injury
Bluegill Sunfish	~51 kPa	>51 kPa
Fall Chinook Salmon	91-99 kPa	91-99 kPa
Rainbow Trout	91-99 kPa	91-99 kPa

Table 2 amount of Pressure values at which mortality/injury occurs for three species [2]

Conclusions

It is clear that the amount of the pressure change that the fish encounter during swimming through a tidal turbine does not significantly endanger the life of the species. There are some locations where the pressure gradient is large but as those zones are located very close to the leading edge of the turbine, the fish going through those areas are likely to be damaged by the impact force from the blade rather than the pressure change. Studies on the interaction of fish and hydroelectric dams [1] showed that the main issue is the head difference before and after the turbine which this is not the case in the TSTs. The effect of cavitation and turbulence require further study, but appear to be negligible. Cavitation happens very close to the blade and turbulence causes some disorientation for the fish.

Acknowledgements:

The Authors wish to acknowledge the financial support of the Crown Estate, Welsh Assembly Government, the Higher Education Funding Council for Wales, the Welsh European Funding Office, and the European Regional Development Fund Convergence Programme.

References:

1. Cada, G.F., *Shaken, Not Stirred: The Recipe for a Fish-Friendly Turbine*, E.S. Division, Editor. 1997, Oak Ridge National Laboratory: Oak Ridge.
2. Becker, J.M., C.S. Abernethy, and D.D. Double, *Identifying the Effects on Fish of Changes in Water pressure during Turbine Passage*, in *Hydro Reveiw*. 2003, HCI.
3. Malki, R., et al. *A Numerical Investigation into Tidal Stream Turbine Wake nDynamics and Device Performance in Non-Uniform Flow* in *International Offshore and Polar Engineering Conference* 2012. Rhodes, Greece: International Sciety of Offshore and Polar Engineering.