PAPER • OPEN ACCESS

Biomechanics of fish swimming in a hydrokinetic turbine wake

To cite this article: M M Macias *et al* 2020 *IOP Conf. Ser.: Mater. Sci. Eng.* **876** 012008

View the article online for updates and enhancements.

Biomechanics of fish swimming in a hydrokinetic turbine wake

M M Macias[1], L Lafuente[2] and A C P Brasil Junior[1]

[1]Mechanical Engineering Department, Faculty of Tecnology, Universidade de Brasilia, 70900190 Campus Darcy Riberio, Brazil
[2]Mathematical Department, Faculty of Engineering, Universidad de Cádiz, 11519 Puerto Real, Spain

marianela@unb.br

Abstract. In the present paper, the assessment of the hydrodynamics of the fish swimming in undisturbed and altered wake-flow is carried out identifying hydrodynamical forces and flow patterns of the fish swimming wake. URANS approach with k-ω/SST turbulence model are employed combining fish and turbine in the same simulation. Fish motion is realized using dynamically adaptive mesh. The actuator line method is employed to induce the wake of a hydrokinetic turbine, which is a simplified method that requires lower computational cost than full geometry simulations. This work brings a new numerical approach involving fish and turbine wake highlighting that fish swimming in the wake presents higher thrust forces than in the undisturbed flow, due to the x-component to velocity in the wake is lower than free flow velocity.

1. Introduction

Nowadays, hydrokinetic energy appears how a technological alternative in the context of sustainable energy. Based on the same principles as wind energy, hydrokinetic devices can produce electricity extracting kinetic energy contained in rivers, estuaries and, channels. Hydrokinetic turbine projects are growing considerably in the last decade however this technology is considered in the pre-commercial stage [1]. Environmental impact studies are a relevant issue yet in development. In literature are described main causes affecting aquatic life due to hydrokinetic devices installation such as sediment suspension, alteration of river hydraulic regimes, noise, electromagnetic fields, fish collisions with blades turbine, toxicity of paints and other chemicals and the installation of several hydrokinetic devices in arrangement [2,3]. Nevertheless, there are not found studies in the literature about hydrodynamic interaction between fish and turbine.

The present paper aims to study fish swimming biomechanics under altered flow conditions similar to hydrokinetic wake topology. Firstly, a numerical methodology is developed to promote tuna fish (*Thunnus atlanticus*) model swimming in undisturbed flow promoting a reliable description of the vortex wake and the computation of hydrodynamic forces. Secondly, the methodology established is employed to carry out analysis of tuna fish (under the same kinematic parameters) swimming in the wake of a hydrokinetic turbine. Transient forces and wakes are computed and compare in both cases, without and with turbine. Simulations were realized in the open-source software OpenFOAM® using the dynamical mesh adaptive tool to simulate fish motion and in the framework of the turbulence, were employed a URANS approach and the k-ω/SST closure turbulence model. Finally, the turbine wake is induced using actuator line model simulations.

2. Numerical methodology

2.1. Actuator Line Method

The Actuator Line Method [4] is a simplified methodology combining Navier-Stokes equation with the Blade Element momentum method (BEMM) to compute the flow around a turbine, calculating wake-flow and hydrodynamical forces on the blades. ALM not employs rotor geometry, blades are considered lines discretized as points and each point represents a blade section. Hydrodynamic forces are computed from an iterative process where the flow solved by Navier-Stokes equations provided information about attack angle and velocity of the flow which are employed in BEM method calculating lift and drag forces on blades (inputting 2D lift and drag forces and airfoil geometry). After all, forces are projected on the background Cartesian grid as body-force field using a three-dimensional Gaussian function and incorporated in the Navier-Stokes. This method is simulated from OpenFOAM® using the library *turbinesFOAM* [5].

In this work, the hydrokinetic turbine is a horizontal axis turbine HK10 designed by the Energy and Environmental laboratory at the University of Brasilia in the context of the P&D project AES Tiete HYDROK. The rotor has 4 blades and 2.2 m of diameter, blade sections are NACA4412 profiles. The operating point employed in the simulations is defined by the tip tip speed ratio, $TSR = \omega R / U$, and the free flow velocity, $U = 2.5\,m/s$ and, rotation velocity, $\omega = 35\,rpm$. For more information about the ALM simulations see [6] where a validation case is presented. The ALM reproduces the three-dimensional structures expected in a wake turbine and therefore, we use this simplified method to induce the wake-flow for the fish swims in the altered flow.

2.2. Fish geometry, kinematics and force computation

Tuna fish geometry is the same as analysed by [7]. The fish body and caudal fin were generated from elliptical and airfoil cross-sections and envelope equations provided by the referenced study, see Fig. 2. Fish swims using the body/caudal fin (BCF) description under carangiform mode. Movement is characterized by undulatory waving motion of its flexible body from head to fin tail that generates a propulsive thrust force and it is described by sinusoidal equation $h(x,t)$ [8] with variable amplitude $a(x)$ such as

$$h(x,t) = a(x)\sin(k\,x - \omega\,t) \tag{1}$$

$$a(x) = a_0 + a_1 x + a_2 x^2 \tag{2}$$

where $h(x,t)$ is the midline fish displacement, ω tail-beat frequency and k the wake number, being $k = 2\pi / \lambda$ the wavelength of propulsive traveling wave. The coefficients a_0 , a_1 and a_2 are determined experimentally and reported in literature. In this work, the kinematic parameters employed were defined from the study of [7] being $\lambda = 1.675$, $\omega = 13\,rad/s$ and the coefficients $a_0 = 0.0$, $a_1 = 0.14$ and $a_2 = 0.000236\,m^{-1}$. Simulations are carried out considering the fish steady in a constant velocity flow.

Hydrodynamic force acting on fish in *i*-direction is computed from the instantaneous velocity and pressure fields, using an integral of the stress-tensor over the fish surface [9]. It is given by

$$F_i(t) = \int_S (-p n_i + \tau_{ij} n_j) dS \tag{3}$$

where τ_{ij} is the stress-tensor components and n_j is the normal of the fish surface. The force time-averaged value (F_i) is obtained integrating the instantaneous force coefficient in an entire cycle of fish swim. If $F_i = 0$ (x is flow direction) the fish swims in a conditions of dynamical equilibrium between the drag and thrust forces. For positive values the fish accelerates upstream (thrust is greater than drag) and for negative value the effect is opposite.

*2.3. Computational domain, numerical mesh and boundary conditions*

The computational domain employed in the simulations is a rectangular prism with dimensions 4D×4D×7D. The inlet flow surface is placed 2D upstream of the turbine and the fish is 2D downstream to rotor (placed at system origin). In figure 3, the rotor is represented by a red rectangle just to help in understanding. Boundary condition imposed on inlet surface is uniform velocity and 5%turbulence intensity. On the outlet surface, a reference pressure is considered to open flow conditions and the lateral walls are free-slip surfaces. Fish surfaces are assumed to be non-slip boundaries, which move according to equations 2 and 3. Due to in ALM simulations there is no turbine surface, no-slip wall boundary condition has not to be imposed.

Numerical mesh was created in OpenFOAM® using the tool *snappyHexMesh*. The mesh on fish surface was highly refined with multiple layers of prismatic elements, to properly simulate of the boundary layer gradients and to maintain the parameter y^+ close to one, as required by the k – ω/SST turbulence model [10]. A mesh convergence study was carried out to guarantee that results do not depend on mesh discretization for fish swimming in undisturbed flow. For the turbine, a mesh convergence was realized in the work [6]. So, in the simulation combining rotor and fish, the element size was kept ensuring the mesh reliability and the number of elements is close to 4 million.

The time-step used in simulations was 10^{-3} keeping the Courant number minor to the unit. For all simulations, free flow velocity was 0.7m/s and tail beat fish frequency 13 rad/s. The rotor kept the Tip speed ratio value was kept to nominal operating condition of the rotor (TSR=1.6), being free flow velocity 0.7m/s and rotational velocity 1.01 rpm.
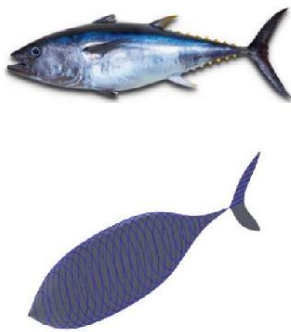

Figure 1. Tuna fish 3D geometry.
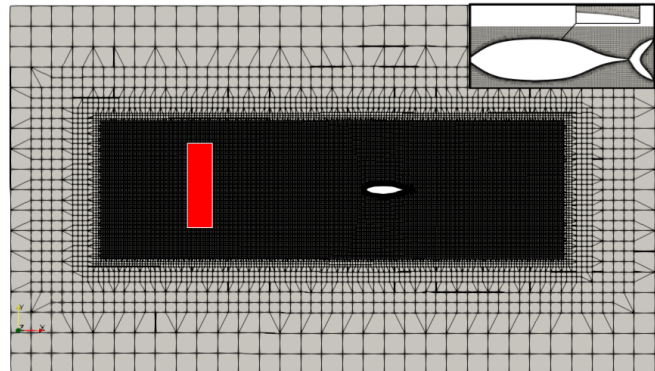

Figure 2. Numerical mesh.

3. Results

Fish wake structure is a relevant issue to understand the resultant propulsion force, evaluating the drag or thrust effect in the framework of the wake topology. In figure 3, dimensionless *x*-velocity component field and *z*-vorticity visualizations allow the comprehension of the fish wake swimming in undisturbed flow and its periodic vortex emission (like a reverse von Kármán vortices). In this situation, the fish is self-propelled and it moves backward flow experiencing a thrust force ($\overline{F_x} > 0$).

Tuna fish presents a jet flow downstream which boosts the fish propulsion, with a vortex sequence organized like positive vortex (counter clockwise - in red colour) above fish symmetry line and negative vortex (clockwise - in blue colour) below that. Time history, for both cases, fish swimming in undisturbed flow and behind the rotor, are presented in figure 4. In both simulations, free flow velocity and fish tail beat frequency are kept equals to understand the effect of the flow turbine on a fish.

From figure 4, it is possible to see that signals begin to differ at time 7s. Around 8.8 s, the black signal corresponding to fish force swimming in the wake-flow achieve maximum values and then, it decreases remaining stable but without returning to the initial values.
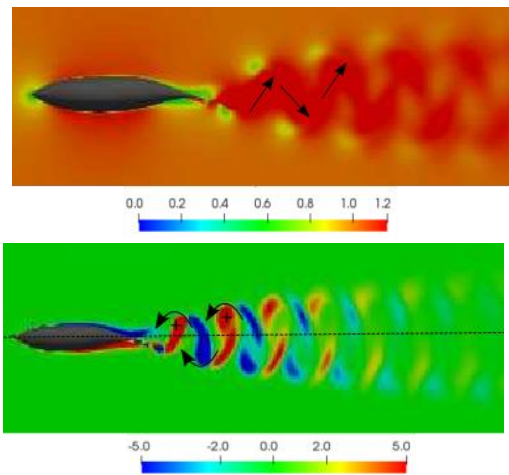
Figure 3. Non-dimensional velocity and vorticity fields, $\left(U/U_\infty\right)$ and $\left(\omega L/U_\infty\right)$ for fish swimming in undisturbed flow.
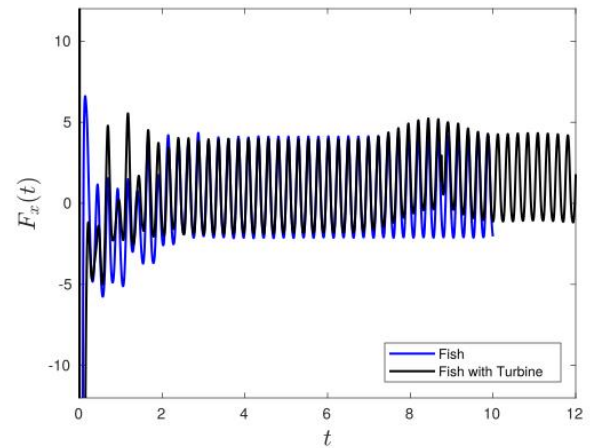
Figure 4. Time history of x-direction forces to fish swimming without and with turbine.

(a)   (b)
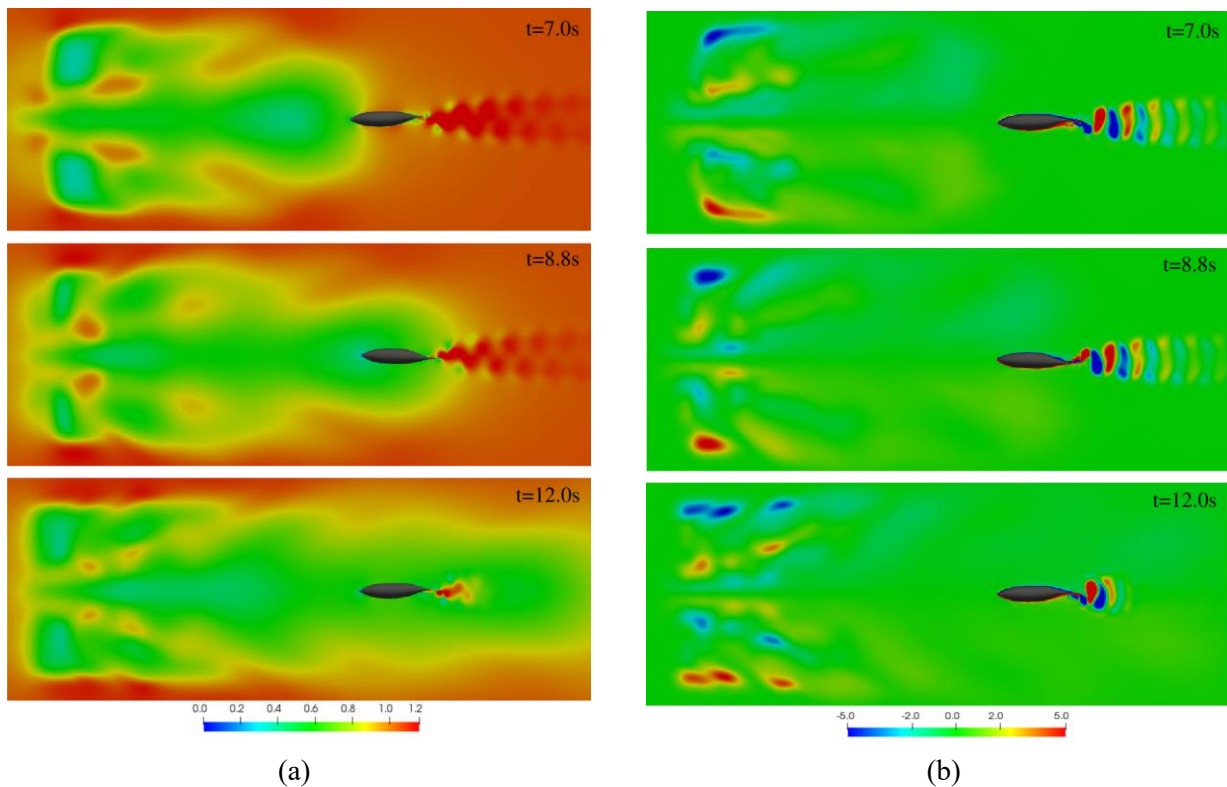
Figure 5. Fish swimming in turbine wake at three different times. (a) Non-dimensional velocity $\left(U/U_\infty\right)$ and (b) vorticity fields, $\left(\omega L/U_\infty\right)$.

From equation 3, mean force in *x*-direction is computed employing 4 periods, at 4s (not turbine interference) and 10s (last times simulated). Results achieving are $\overline{F_x} = 0.63N$ and $\overline{F_x} = 1.30N$,

respectively. For all, fish experiences acceleration in opposite direction to the flow due to thrust-type force ($\overline{F_x} > 0$).

From figure 5, we can be able to relates turbine wake evolution with time history in figure 4. Firstly, we verify that fish find the wake-flow at 7s. From that instant, fish swims at lower velocities and experiences higher forces due to tail beat frequency is kept constant. At instant 8.8 s, where maximum forces values were noted, a region of the fluid with even lower velocity reaches the fish. Last, at higher times, fish is swimming immersed in a permanent wake where force signal oscillates around a mean value. In figure 5, when t=12s, fish wake appears shorter due to it is pushed by the flow of wake turbine but the wake maintains a reversal von Kármán vortex street configuration generating propulsive force.

4. Conclusion

Numerical simulations for tuna fish swimming in an undisturbed flow and an altered flow were presented in this work. The altered flow has the hydrokinetic turbine wake topology and it was induced through simulations by the simplified actuator line method. Results obtained from CFD study allow us to conclude that to both cases (with and without turbine), fish experiences thrust force appearing the wake like a reverse von Kármán vortex street generating a jet flow downstream which boots the fish propulsion. But, fish swimming in the turbine wake (at the same tail beat frequency) presents a higher mean force due to the flow velocity is lower. Although, these results contribute to the understanding of fish swimming in altered flow. It should be necessary to compute the propulsive efficiency at the frequency equilibrium condition and to analyse the vortex effect to conclude fish performance behaviour.

5. References

[1]   Laws N D and Epps B P 2016 Technology, research, and outlook *Renew. Sust. Energ.* **57** pp 1245–1259
[2]   Kartezhnikova M and Ravens T M 2014 Hydraulic impacts of hydrokinetic devices *Renew. Energy* **66** pp 425–432
[3]   Fallon D, Hartnett M, Olbert A and Nash S 2014 The effects of array configuration on the hydro-environmental impacts of tidal turbines *Renew. Energy* **64** 10–25
[4]   Sørensen J N and Shen W Z 2002 Numerical Modeling of Wind Turbine Wakes *J. Fluids Eng.* **124** pp 393
[5]   Yu Z, Zheng X and Ma Q, 2018 Study on Actuator Line Modeling of Two NREL 5-MW Wind Turbine Wakes *Appl. Sci.* **8** p. 43
[6]   Macias M M, Mendes R C F, Brasil Junior A C P and Felamingo T O Hydrokinetic turbine simulations using actuator line method *Proc. of the COBEM 2019: 25th ABCM International Congress of Mechanical Engineering* Uberlandia, MG,Brasil ABCM
[7]   Zhu Q, Wolfgang M J, Yue D K P and Triantafyllou M S 2002 Three-dimensional flow structures and vorticity control in fish-like swimming *J. Fluid Mech.* **468** pp 1-28
[8]   Lighthill M J 1969 Hydromechanics of Aquatic Animal Propoulsion *Annu. Rev. Fluid Mech* **1** pp 413-446
[9]   Borazjani I and Sotiropoulos F 2008 Numerical investigation of the hydrodynamics of carangiform swimming in the transitional and inertial flow regimes *J. Exp. Biol.* **211** pp 1541–1558
[10]   Menter F R 1994 Two-equation eddy-viscosity turbulence models for engineering applications *AIAA Journal, American Institute of Aeronautics and Astronautics* **32** pp 1598-1605