Numerical modelling of fish passage and flow interaction in a hydroturbine

I. Kassanos
School of Mechanical Engineering
National Technical Univ. of Athens
15780 Athens, Greece

V. Sanoudos-Dramaliotis
School of Mechanical Engineering
National Technical Univ. of Athens
15780 Athens, Greece

J.S. Anagnostopoulos
School of Mechanical Engineering
National Technical University Athens
15780 Athens, Greece

Introduction

The recent interest in the development of new hydropower plants, the exploitation of lower head sites for energy production or storage, and the refurbishment of old hydro and pump units, as also the adoption, on the other hand, of stricter environmental terms and restrictions, make necessary the design of new hydroturbines and reversible machines with improved environmental performance.

A significant environmental impact of hydropower plants is the injury and fatality of the fish fauna passing through the hydroturbines. The present work aims to contribute in the research to mitigate this impact, by the development of a computational tool that can be used to assess the fish-friendly performance of hydroturbines, and to improve and optimize their design.

The methodology is based on the simulation of the motion of a solid body through the turbine guide vanes and rotor, during which the hydrodynamic forces acting on the body due to pressure, shear, acceleration and turbulence are calculated and recorded, as also any possible impacts of the body on the blades or hub/shroud surface. The shape and size of the solid body can be determined so as to approximate a real fish species.

At first, the solid body is tracked into a converged flow field in a 2D blade-to-blade domain of a Francis turbine rotor, and the collected data are used to quantify the combined effect of the rotor on the body, based on relevant indexes from the literature. Next, a fluid structure interaction procedure is implemented for the same test case, in order to estimate the differences between the object pathway and the flow streamlines, assuming a passive response of the solid body to the surrounding flow.

Sensitivity tests are carried out in order to assess the effect of the turbine operation parameters (e.g. flow rate), as well as of the fish morphology (e.g. size). Also, the effect of the hydrodynamic design of the rotor (e.g. blades number) is studied. The results are analyzed and processed in order to start formulating a multivariable function that could be used as a quantitative fish-friendliness index for the customized optimum design of hydroturbines for a specific site and fish species

1. Background

The increased complexity of blade passage simulation has led to the development of fish mortality evaluation algorithms based on biological performance indicators derived from the CFD simulation of the turbine's operation. Specifically past attempts to predict fish mortality on a specific turbine environment have focused on identifying the locations and size of potentially hazardous regions (Garrison et al., 2002; Keller et al., 2006; Cada et al., 2006). Limiting the number and volume of such regions was the main objective towards reduction of fish mortality during turbine passage. However this methodology does not take into account the probability of fish passage through those areas.

Minimum pressure threshold is considered a similar criterion guiding the design of turbine with regards to fish friendliness (Brown et al., 2012). While minimum pressure criterion is easy to implement on the design of a turbine it still does not take into account the probability of fish passage through the minimum pressure area of the turbine.

More recent work has led to the development of probabilistic methodologies taking into consideration the probability of a fish to be exposed in an injury mechanism. Specifically, the Pacific Northwest National Laboratory (PNNL) has developed the Biological Performance Assessment (BioPA), a design methodology utilizing biological performance indicators for injury and mortality calculated on data from a CFD model of a proposed turbine design. In order to calculate the probability of fish mortality, those biological performance indicators are calculated on streamlines on a CFD simulation that model potential fish pathways through the turbine. By using an appropriate sample of these streamlines it is possible to calculate the fish survival rate, taking into consideration different mortality criteria along each streamline. This way the total turbine biological performance can be assessed. (Richmond et al., 2014).

It is obvious that the streamline tracing method greatly improves the quality of the simulation results regarding the turbine's true biological performance. It relies, though, in the general assumption that fish pathways through the turbine are identical or can be modeled with the numerically solved streamlines, hence biological indicators are calculated on those turbine streamlines. In order to overcome this assumption different modeling techniques have been developed. Specifically, with the method of Lagrangian particles a fish is simulated with spherical Lagrangian particles with mass, adding additional physical realism compared to streamlines albeit at increased computational cost. Lagrangian particles are not passively advected through the flow but instead, the particle velocity is determined by solving an ordinary differential equation that includes fluid forces acting upon the particle such as drag, pressure gradients, virtual mass effects and gravity when fish are made non-neutrally buoyant (Richmond and Romero-Gomez, 2014).

Towards greater fidelity on the fish true pathline, the method of composite discrete element modeling –an improvement of the Lagrangian particle method– is used. In this method fish is simulated with a series of discrete spherical sub-particles. The motion of the spherical sub-particles is determined by the equation of motion of particles along with the angular momentum balance equations, leading to a six degree of freedom for the composite body. This approach takes into account the target fish length compared to the Lagrangian particle simulation, which affects the simulations fidelity. The method, though, does not simulate the fluid forces on the fish body.

The abovementioned methodologies present certain advantages and disadvantages, rendering each appropriate for certain applications. It is for instance safe to assume that on a large scale turbine, where fish are very small compared to the turbine size, the streamline tracing method leads to similar results with the Lagrangian method. But if the fish size is comparable with the turbine size, the fish mass, density and shape should be taken into account, the fluid forces acting on the fish should be modelled, while the effect of the fish passage on the flow field may be also included.

The present methodology is based on the simulation of the fish passage through a reaction turbine, by modelling the fluid forces acting on a solid object that has mass and dimensions. The biological performance indicators can be recorded along the calculated pathway, offering a more realistic evaluation of the conditions encountered by the passing fish, and allowing for an assessment of the impact of the turbine on their mortality.

2. Numerical methodology

The 2D computational domain considered here is a spiral casing with a simple runner model consisting of 7 or 9 blades (Fig. 1a). In order to simulate the fish passage, a cylindrical object with thickness c and length l was considered (Fig. 1b) and released from different locations in the inlet section of the spiral casing (e.g. points A, B and C in Fig. 1a). The commercial code Ansys Fluent™ was used to solve the incompressible Navier stokes equations. The k-ε realizable turbulence model with scalable wall functions was used, while the object movement was simulated using the dynamic mesh model. In order to avoid re-meshing of the domain as the object moves through the flow field, the Overset technique is implemented [8]. Based on this technique, the domain is discretised using a background mesh, on which individual component meshes are overlain. In the present application two component meshes, one for the runner and one for the object/fish, were considered (Fig. 1b).

As the object moves through the flow field, the location of the overset interface and at the same time, the active cells, where the flow equations are computed, are updated. The information between the meshes is passed by interpolating the flow variables across the interface. In this way it is possible to avoid re-meshing, while at the same time the grid quality is maintained. Using this approach, the final mesh consisted of 214,143 cells, of which 32,917 were used for the runner and 1,046 cells for the object/fish.

A second order in space and first order discretization in time was employed, while the flow equations were solved in a coupled manner. The velocity was prescribed at the inlet of the domain, while a zero pressure boundary conditions was applied at the outlet. Finally, a no-slip boundary condition was applied on all walls. In order to ensure stability of the solution, the flow filed is initialized using a steady state solution and the Moving Reference Frame (MRF) model [9]. After an initial steady state solution is obtained, the MRF model is activated and the runner is set to rotate at the prescribed rotational speed. Once convergence is achieved the transient simulation is activated, with the object fixed in the initial position. The solution is evaluated with a time step equivalent to a runner rotation of 2 degr/step. Once the flow behind the object is stabilized, the object movement is initiated by solving the equations describing the object motion. The motion of the object was described as a solid body motion using the 6-DOF model [10]. During this solution process, the flow equations are not updated, and therefore the motion of the object is dictated by the initial field. The fish/body retains mass and density properties, while custom functions are used to simulate its possible rigid body impacts with the turbine internal walls or runner blades.

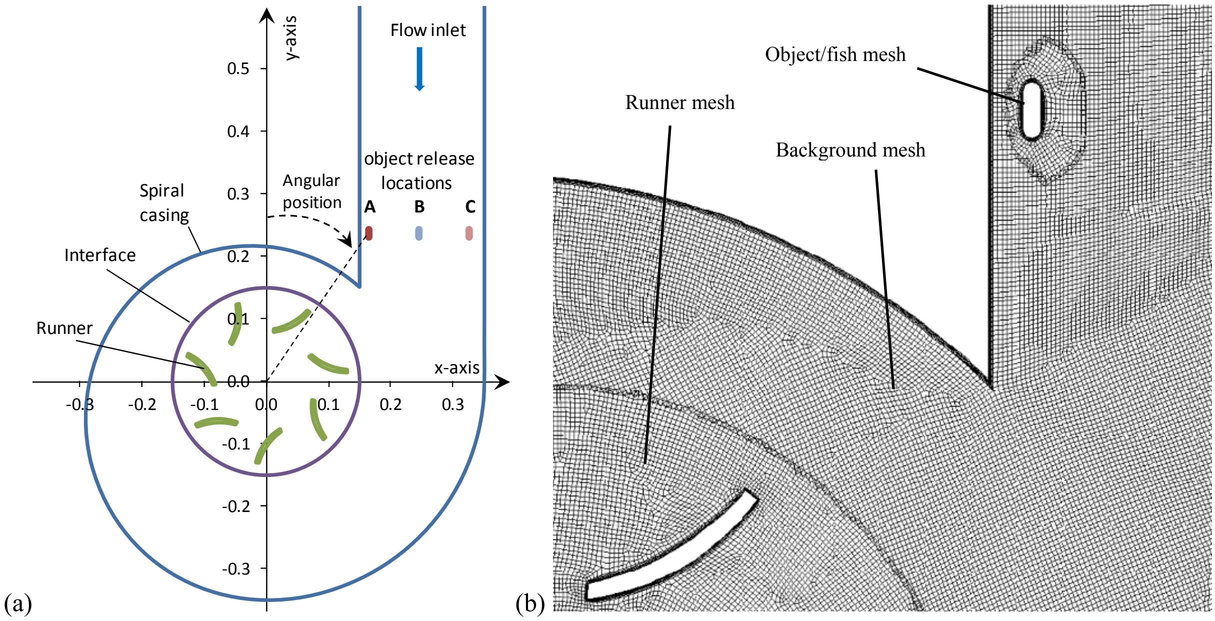


Fig. 1. The computational domain (left), and details of the overset meshes (right).

3. Results and Discussion

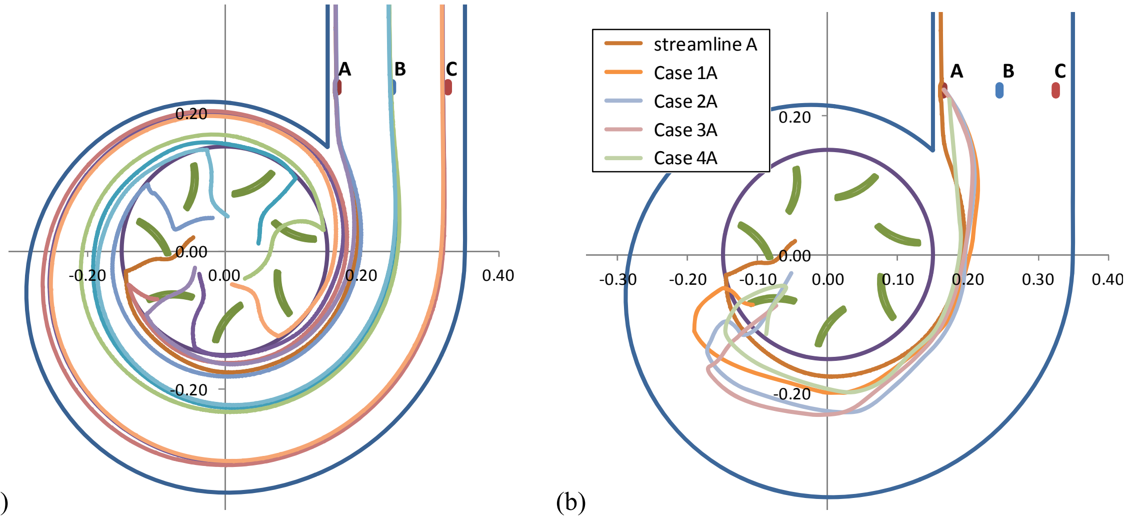
A simplified 2D domain of a Francis turbine without wicket gate is considered for first application and evaluation of the above numerical methodology (Fig. 1). The turbine dimensions correspond to laboratory model turbine, and the nominal operating conditions were: Flow rate 0.03 m^3/s, hydraulic head 2.5 m, and rotation speed 333 rpm.

The present parametric simulations of the object/body motion in the turbine are carried out for two different flow rates and two object shapes with different aspect ratio. Also, two runners, one with five and one with seven blades are examined. These test cases are tabulated in Table 1, below. For each of them, the object/fish can be released from three different positions in the inlet section (A, B, C, Fig. 2). The corresponding stream lines of the converged flow field for the cases C1 to C4, that are passing through these release points, are drawn in Fig. 2a. It can be seen from this figure that the outer streamlines (point C) are quite similar, whereas more pronounced are the differences of the inner streamlines (point A).

The same is valid for the corresponding trajectories of the object/fish motion, which are drawn in Fig. 2b. But the differences between cases C1 to C4 are quite more enlarged, while all deviate considerably from the fluid streamlines. This behaviour is the result of the present modelling, by which the object has not only mass but also dimensions, and hence it may also rotate along its pathway due to the action of pressure and shear forces. This can be observed in the indicative pictures of the object position and orientation along its pathway through the turbine, given in Fig. 3. In this particular trajectory, the object impacts on a rotating blade and its trajectory calculation terminates.

Table 2. The various test conditions examined numerically.

Cases	Flow rate (% Q_N)	Number of blades	Object shape (l:c)
C1 (reference)	High 100	7	Thin 4:1
C2	**Low 70**	7	Thin 4:1
C3	High 100	**9**	Thin 4:1
C4	High 100	7	**Thick 4:2**



(a) (b)

Fig. 2. Flow streamlines through points A, B and C (a); and object pathways released from position A (b).

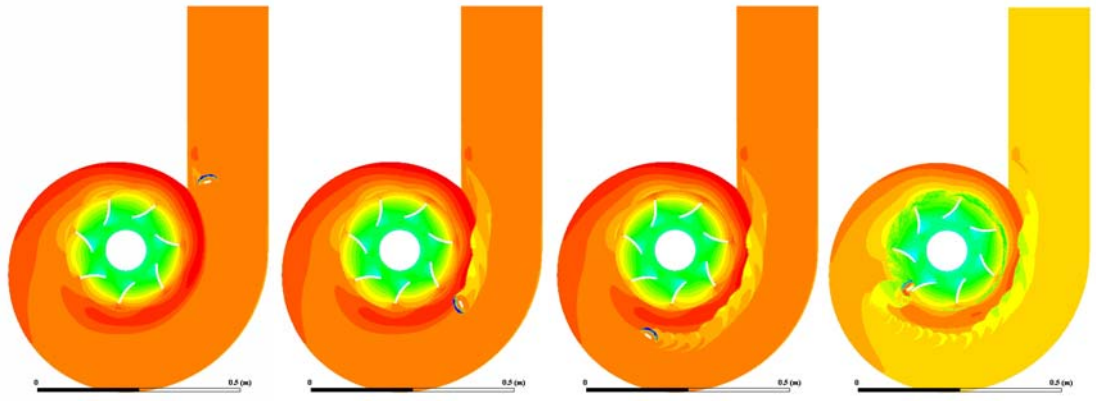


Fig. 3. Indicative snapshots from the simulated motion of an object/fish motion through the turbine flow field.

The differences between the object pathway and the flow streamlines observed in Fig. 2b, result in different behaviour of the object, regarding the imparted velocity and acceleration along its pathway, as can be shown in Fig. 4, where the angular position of the trajectories is defined in Fig. 1a. The object velocity components, u and v (Fig. 4a and 4b, respectively), exhibit remarkable deviation from the corresponding fluid velocities along a streamline passing through the object release position (position A). This, in turn, results in even larger differences regarding the corresponding acceleration components (Fig. 4c and 4d). The latter becomes quite higher for a fluid particle than for the solid object as the trajectories approach the runner inlet (angular position > 180 deg). Therefore, the use of the fluid streamlines to assess the possible impact of the turbine on the survivability of the passing fishes cannot produce realistic results in the present case.

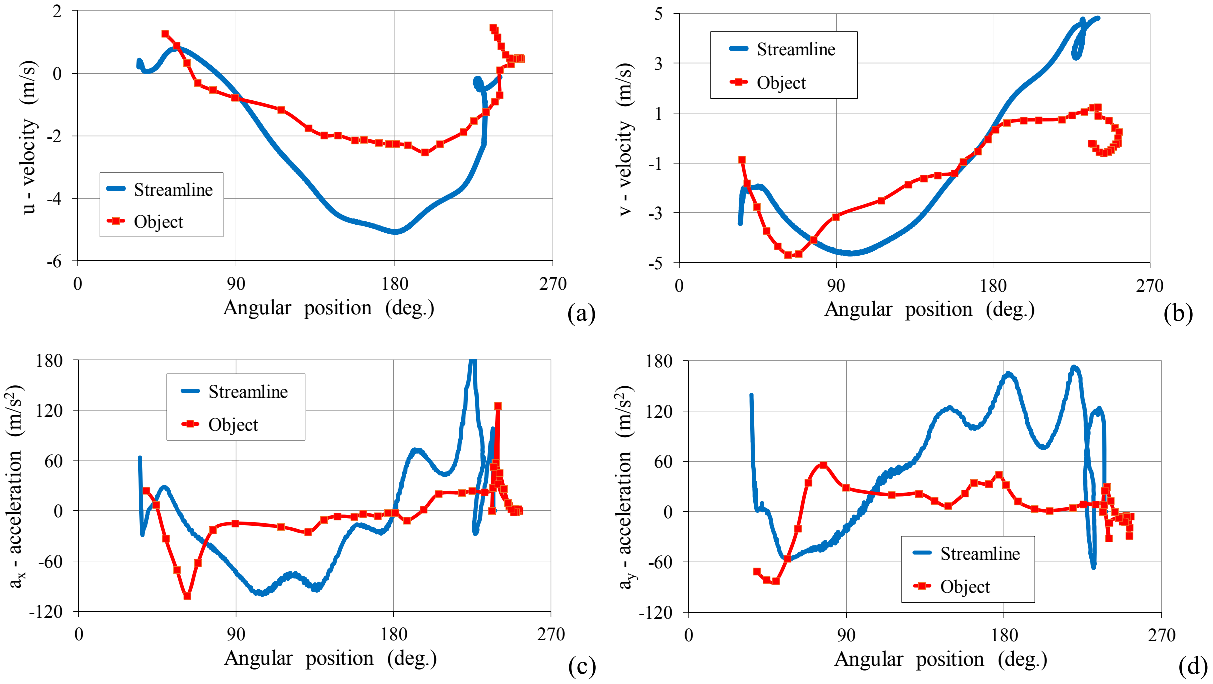


Fig. 4. Comparison between fluid and object velocity components (a, b), and acceleration components (c, d).

4. Conclusions

The overset mesh technique was implemented in a numerical methodology that simulates the pathway of a solid object/fish. The new method was tested for a simplified 2D reaction turbine geometry. It was found that the object trajectories deviate from the calculated flow streamlines and follow a different route through the turbine casing and runner. As a result, the object acquires different velocities and acceleration than the fluid particles.

Consequently, the present method seems capable to produce more realistic results than a simplified approach using a Lagrangian or streamline method. However, its computational demands are much higher, and hence further research is required to reduce the numerical evaluation time, in order to be used for complete statistical analysis of the impact of the turbine design parameters on the passing fish survivability.

Acknowledgment: This research has been co-financed by the European Union and Greek national funds through the Operational Program Competitiveness, Entrepreneurship and Innovation, under the call RESEARCH – CREATE – INNOVATE (project code: T1EDK-01334, Oct. 2018 – Oct. 2021)

European Union
European Regional Development Fund

HELLENIC REPUBLIC
MINISTRY OF ECONOMY & DEVELOPMENT
SPECIAL SECRETARY FOR ERDF & CF
MANAGING AUTHORITY OF EPAnEK

EPAnEK 2014-2020
OPERATIONAL PROGRAMME
COMPETITIVENESS
ENTREPRENEURSHIP
INNOVATION

ΕΣΠΑ 2014-2020
ανάπτυξη - εργασία - αλληλεγγύη
Partnership Agreement
2014 - 2020

Co-financed by Greece and the European Union

References

1. **Brown, R.S., Ahmann, M.L., Trumbo, B., Foust, J.,** "Fish-protection: cooperative research advances fish-friendly turbine design", *Hydro Review* 31, 1–6, 2012.
2. **Richmond, M.C., Serkowski, J.A., Ebner, L.L., Sick, M., Brown, R.S., Carlson, T.J.,** "Quantifying barotrauma risk to juvenile fish during hydro-turbine passage", *Fisheries Research*, vol. 154, pp. 152-164, 2014.
3. **Garrison, L.A., Fisher, R.K., Sale, M.J., Cada, G.F.,** "Application of biological design criteria and computational fluid dynamics to investigate fish survival in Kaplan turbines", In: *HydroVision 2002*, HCIPublications, pp.1–11, 2002.
4. **Cada, G., Loar, J., Garrison, L., Fisher, R., Neitzel, D.,** "Efforts to reduce mortality to hydroelectric turbine-passed fish: locating and quantifying damaging shear stresses", *Environmental Management* 37, 898–906., 2006.
5. **Keller, M., Sick, M., Grunder, R., Grafenberger, P.,** "CFD-based assessment of fish- friendliness of the time dependent flow field in a Kaplan runner". In: *HydroVision 2006*, HCI Publications, 2006.
6. **Richmond M.C. and Romero-Gomez**, **P.,** "Fish passage through hydropower turbines: Simulating blade strike using the discrete element method", *IOP Conf. Ser.: Earth Environ. Sci.* 22, 2014.
7. **Romero-Gomez, P., Michelcic, J., Lang, M., Weissenberger, S.,** "The integration of fish-friendliness metrics in industry practices for turbine design", *20th Intern. Seminar on Hydropower Plants*, Vienna, Austria 14-16 November, 2018.
8. **Steger JL, Dougherty FC, Benek JA**. A Chimera grid scheme. In: Ghia KN, Ghia U, editors. *Advances in grid generation*. ASME FED,Vol. 5, 1983.
9. **Luo, J.Y., Issa, R.I., and Gosman, A.D.,** "Prediction of Impeller-Induced Flows in Mixing Vessels Using Multiple Frames of Reference", *IChemE Symposium Series*, 136, pp. 549-556, 1994.
10. ANSYS Fluent Theory Guide, ANSYS, Inc., 275 Technology Drive Canonsburg, PA 15317, November 2013.

The Authors

Ioannis Kassanos graduated in Mechanical Engineering from the National Technical University of Athens, Greece, and has and MSc in Energy Conversion and Management from the University of Nottingham. He works with a UK hydroturbines manufacturer and he is completing his Ph.D. in numerical design of reaction hydroturbines at the Laboratory of Hydraulic Turbomachines, NTUA. He is experienced in CFD simulation, performance prediction, fluid structure interaction and design optimization of reaction turbines and their components, as well as in field measurements and Laboratory testing of hydraulic turbines.

Vassos Sanoudos-Dramaliotis graduated in Mechanical Engineering from the National Technical University of Athens, Greece. He worked in the field of hydropower in the private sector and as a consultant in the fields of machine learning in a London-based fund, government and public services in Price Waterhouse Coopers and IC Engines in the thermo-fluids department of Imperial College, London. He is currently completing his Ph.D. in the field of reaction hydraulic turbines at the Laboratory of Hydraulic Turbomachines, NTUA.

Ioannis S. Anagnostopoulos graduated in Mechanical Engineering from the National Technical University of Athens, Greece, and received his Ph.D. in Computational Fluid Mechanics from the same University. He worked for several years as post-doctoral researcher in the NTUA and as R&T consultant in the private sector, where he has been involved in feasibility studies for various industrial innovations. He has participated in more than 40 research projects, and has more than 100 scientific publications in international journals and conferences. Also, he has developed a number of advanced computer codes for the simulation of various fluid flow mechanisms in industrial applications, as well as for modelling and optimisation of the design and operation of hydroelectric and hybrid energy systems with pumped storage. He is Professor in the School of Mechanical Engineering NTUA and Director of the Laboratory of Hydraulic Turbomachines, and his current research activities include numerical and experimental flow analysis and hydrodynamic design in pumps and hydroturbines.