COMPUTATIONAL MECHANICS 2023
38th conference with international participation
Srní
October 23 - 25, 2023

On tank hydrodynamics of recirculating aquaculture systems: Computational fluid dynamics modeling and its validation

Š. Papáček[a], M. Schuster[b], J. Urban[a]

[a] *University of South Bohemia in České Budějovice, FFPW, Institute of Complex Systems, 373 33 Nové Hrady, Czech Republic*
[b] *Research and Testing Institute Plzeň Ltd., Tylova 1581/46, Plzeň, Czech Republic*

1. Introduction

Recirculating aquaculture systems (RAS) form an important subset of aquaculture systems, providing about half the fish for human consumption worldwide. The demand for fish is rising, but fisheries are not expected to grow due to fully or over-exploited fish stocks. Therefore, there exists a strong demand in the public and private sectors of aquaculture research, which has to work toward sustainable production of high-quality fish with reduced environmental impact [3]. In the frame of the AQUAEXCEL (Aquaculture infrastructures for excellence in European fish research) project[1] we aim to bridge the gap between the the scientific community and the fish industry through the stimulation of problem-based research and enhanced knowledge transfer.

As follows, we focus on developing CFD (Computational Fluid Dynamics) based methodology for numerical simulation and further scale-up of aquaculture systems. The principal goal is to present a CFD-based simulation related to RAS tank hydrodynamics. As far as we know, only a few works are related to this area; see [7] for general agro-environmental application and [5] for a study on RAS. Here, as a proof-of-concept, we provide the steady-state flow fields of the conical-cylindrical tank used in the Hellenic Centre for Marine Research (HCMR) for larvae rearing [8].

Moreover, knowing that the numerical model validation is the paramount issue within CFD-based methodology, we further compare the experimentally measured and numerically simulated profiles of the flow field within tanks. Eventually, having a reliable method to simulate the fluid flow within the RAS tanks, we can address two important features of aquaculture tank design and operation, i.e., (i) to maximize the capability of the system for self-cleaning, and (ii) to ensure the optimal rearing conditions (including the fish exercising).

While the question of whether the tank will effectively clean or not was already extensively treated, see [9, 10] and references within there, the topic of fish exercising, which can be viewed as a sub-case of fish wellbeing is only emerging [4, 6, 11]. Here, we place the CFD-based methodology to solve the above-proposed problems once being precisely formulated.

---

[1] AQUAEXCEL3.0 is a research infrastructure project funded under the EU Horizon 2020 research and innovation program (grant agreement No. 871108) coordinated by the French National Institute for Agricultural Research (INRA). It aims to support the sustainable growth of the European aquaculture sector. AQUAEXCEL3.0 integrates a large group of leading European aquaculture research facilities (22 partners from 13 countries) and aims to advance aquaculture research and innovation in Europe.

2. Fluid-dynamic model of RAS

As follows, we model only one incompressible Newtonian liquid phase (fresh or sea water) in RAS.[2] Therefore, the classical system of Navier-Stokes equations and the continuity equation are used as fluid-dynamic model

$$\frac{\partial u}{\partial t} + (u \cdot \nabla)u = f - \frac{1}{\rho}\nabla p + \nu \nabla^2 u, \quad \nabla \cdot u = 0 \quad \text{in} \ (t_0, T) \times \Omega \tag{1}$$

with corresponding boundary conditions (reflecting the tank geometry) on $(t_0, T) \times \partial\Omega$ and initial conditions (reflecting the tank operating conditions) in Ω, and where $u,\ p,\ f,\ \rho$ and ν denote the fluid velocity, the pressure, the body forces, fluid density, and kinematic viscosity, respectively.

Well-established methods exist to solve (1). Nowadays, the commercial CFD codes prefer the finite volume approach (FVM) to obtain the discretized form of (1) than other approaches like FDM or FEM (finite difference or finite element methods, respectively), especially with unstructured grids. The system of discretized equations (very large system of linear algebraic equations) is solved, usually iteratively, to find the values of velocities and pressures in all grid points. The coupled set of governing equations (1) is discretized in time for steady and unsteady calculations. In the steady case, it is assumed that time marching proceeds until a steady-state solution is reached.

The conical-cylindrical tank geometry used in this work is described in [8]. Among the three tank volumes (small of 40 liters, medium of 0.5 m^3, and big of 2 m^3), the 0.5 m^3 tank for fish larvae tank was chosen. The mean hydraulic retention time (HRT) was set to 120 minutes, corresponding to the volumetric flow rate of 0.25 m^3/hour, and the inlet velocity was 0.35 m/s. Some details about the inlet and outlet arrangement were communicated personally. The tank 3D representation, made by the CFD code ANSYS Fluent [2], is depicted in Fig. 1 (left). Computational mesh of 0.46×10^6 so-called polyhedral elements converted by merging from the originally 1.9×10^6 tetrahedral elements (besides the reduction of the total number of elements, the new polyhedral mesh is showing improved orthogonal quality and reduced skewness) was

---

[2] The simulation of solid particle movement was treated in our study [12], while the study about the incorporation of fish swimming (and respiration) is still under development.

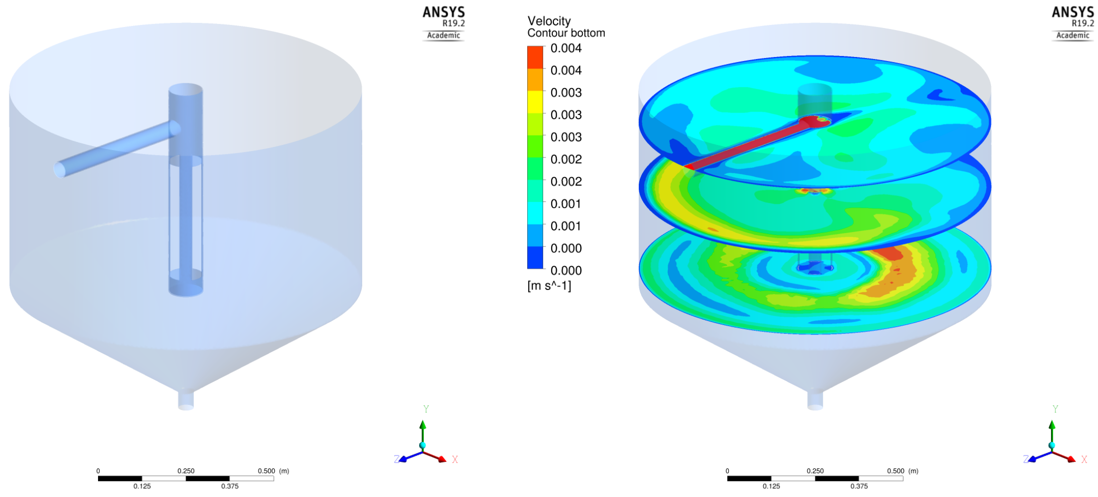

Fig. 1. (Left) Tank geometry of the RAS for rearing fish larvae at HCMR Heraklion, Greece. (Right) Velocity contours in three horizontal planes (40, 400, and 800 mm above the tank bottom)

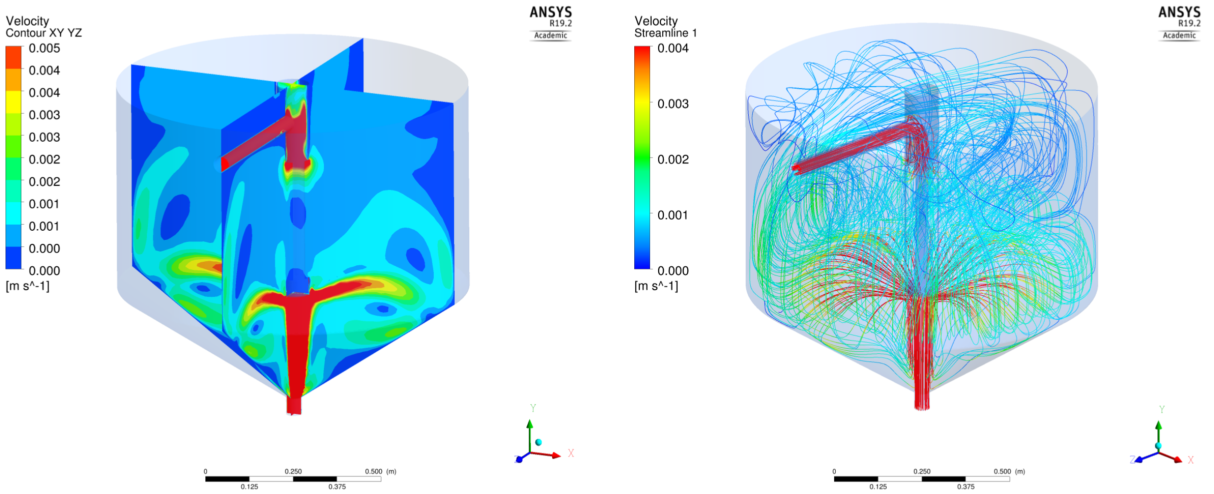

Fig. 2. (left) Velocity contours in two perpendicular axial cross-sections. (right) Streamlines illustrating the flow pattern in the HCMR tank

used for the velocity flow field calculation in the whole domain. As an illustration of CFD results, the velocity contours in three horizontal planes are depicted in Fig. 1 (right), while the velocity contours in two perpendicular axial cross-sections are shown in Fig. 2 (left) and the streamlines are plotted in Fig. 2 (right).

3. Validation of CFD simulation

The validation of the CFD model is defined as the process of determining the degree to which the model is an accurate representation of the real world from the perspective of the intended uses of the model [1]. Validation basically depends on directly comparing CFD-computed results and measured experimental results. Fluid velocity measurements were performed using a Vectrino velocimeter, a high-resolution acoustic velocimeter used to measure three-dimensional velocity in various applications from the laboratory to the ocean. The measurement technology is based on the coherent Doppler processing, having a range of ± 1 cm s^{-1} with an accuracy of ± 0.5 % of the measured value or ± 1 mm s^{-1}. The sampling volume is at a distance of 5 cm from the probe, with a diameter of 6 mm and a height of 7 mm. A specially prepared construction on the top of the tanks allowed the accurate positioning of the sensor in the tank to perform the measurements. The 0.5 m^3 tank was divided into three layers, where 12 measurements were taken at the first two layers and eight measurements at the third layer.

Afterward, the CFD results for the liquid phase velocity in the bottom, the medium, and the surface plane were used to calculate the average velocities in these planes, see Fig. 1 (right). Finally, these results were compared with the experimentally measured values published in [8] (see Table II on page 1 279).

4. Conclusions

Our work is a step towards using CFD for aquaculture tank design, optimization, and the scale-up of a laboratory system. We argue that the CFD-based methodology is an economically feasible tool, making it possible to conduct simulation studies on a desktop PC. Our CFD validation study has shown comparable results to real-world measurements, even though we do not assume the measurement is more accurate than the CFD results because the validation methodology only asserts that experimental measurements are the only true reflection of reality and that the estimation process for error and uncertainty must be effectuated on both sides [1]. Even though CFD calculations may not be sufficiently exact, the degree of error is within reasonable bounds for engineering purposes. Thus, CFD analysis can become the standard tool for RAS design and operation, reducing research and engineering development costs.

Future work can focus both on biological and computational sides; i.e., we aim first to determine a simple description of fish swimming inside RAS and then to implement it to a suitable CFD code.

For instance, the description of the influence of the fish swimming on the flow pattern, see, e.g., [5, 9] and the quantification of fish preferences [11], in order to set up the design criteria in the optimization procedure with CFD runs embedded.

Acknowledgments

This work was supported by the MEYS of the Czech Republic – projects CENAKVA (No. CZ.1.05/2.1.00/01.0024), CENAKVA II (No. LO1205 under the NPU I program), by AQUAEXCEL, and by AQUAEXCEL3.0 research infrastructure project funded under the EU Horizon 2020 research and innovation programme (grant agreement No. 871108).

References

[1] AIAA-G-77 – Guide for verification and validation of computational fluid dynamics simulation, American Institute of Aeronautics Astronautics, 1998.

[2] ANSYS Fluent. http://www.ansys.com/Products/Fluids/ANSYS-Fluent

[3] AQUAEXCEL^{2020} – AQUAculture infrastructures for EXCELlence in European fish research towards 2020. https://aquaexcel2020.eu/

[4] Davison, W., Herbert, N. A., Swimming – enhanced growth, In: Swimming physiology of fish: Towards using exercise to farm a fit fish in sustainable aquaculture, (eds.) A. P. Palstra, J. V. Planas, Berlin, Springer, 2013, pp. 172-202.

[5] Gorle, J. M. R., Terjesen, B. F., Mota, V. C., Summerfelt, S., Water velocity in commercial RAS culture tanks for Atlantic salmon smolt production, Aquacultural Engineering 81 (2018) 89-100.

[6] Hanák, J., CFD simulation of flow in fish tanks, Master thesis, CTU Prague, Prague, 2016.

[7] Lee, I. B., Bitog, J. P. P., Hong, S.-W., Seo, I.-H., Kwon, K.-S., Bartzanas, T., Kacira, M., The past, present and future of CFD for agro-environmental applications, Computers and Electronics in Agriculture 93 (2013) 168-183.

[8] Lika, K., Pavlidis, M., Mitrizakis, N., Samaras, A., Papandroulakis, N., Do experimental units of different scale affect the biological performance of European sea bass Dicentrarchus labrax larvae? Journal of Fish Biology 86 (4) (2015) 1271-1285.

[9] Masaló, I., Oca, J., Influence of fish swimming on the flow pattern of circular tanks, Aquacultural Engineering 74 (2016) 84-95.

[10] Oca, J., Masalo, I., Flow pattern in aquaculture circular tanks: Influence of flow rate, water depth, and water inlet & outlet features, Aquacultural Engineering 52 (2013) 65-72.

[11] Palstra, A. P., Mes, D., Kusters, K., Roques, J. A. C., Flik, G., Kloet, K., Blonk, R. J. W., Forced sustained swimming exercise at optimal speed enhances growth of yellowtail kingfish (Seriola lalandi), Frontiers Physiology 5 (2015) No. 506.

[12] Papáček, Š., Petera, K., Císař, P., Stejskal, V., Saberioon, M., Experimental & computational fluid dynamics study of the suitability of different solid feed pellets for aquaculture systems, Applied Sciences 10 (19) (2019) No. 6954.