On the Optimization of Recirculated Aquaculture Systems

Štěpán Papáček[1(✉)], Karel Petera[2], Ingrid Masaló[3], and Joan Oca[3]

[1] Institute of Complex Systems, University of South Bohemia in České Budějovice,
FFPW, CENAKVA, 373 33 Nové Hrady, Czech Republic
`spapacek@frov.jcu.cz`
[2] Faculty of Mechanical Engineering, Czech Technical University in Prague,
Technická 4, 166 07 Prague 6, Czech Republic
[3] Departament d'Enginyeria Agroalimentária i Biotecnologia, Universitat Politècnica
de Catalunya BarcelonaTECH, C/Esteve Terrades 8, 08860 Castelldefels, Spain

Abstract. The improved design of recirculated aquaculture systems (RAS) is needed facing the demand for increased fish production as well as increased concern of fish wellbeing. Here, we make a step towards using of computational fluid dynamics (CFD) for the optimization of fish tanks. The proposed CFD based methodology allows the modeler (the designer) to manipulate both the tank geometry and operating conditions, in order to solve a multicriteria optimization problem. The individual objective functions are quantifying multiple criteria, dealing mainly with the rearing conditions for fish (a determined average velocity and low velocity variance), fast biosolids removal and the cost of energy and place. As an example of our methodology, we present a study involving the CFD analysis of four different RAS: (i) the circular, (ii) the octagonal, (iii) the square, and (iv) the rectangular multivortex RAS, either with or without additional plate baffles between two water inlets. Based on the two-dimensional description of the flow field within each RAS configuration, the set of efficient solutions for RAS is inferred.

Keywords: CFD · Recirculated aquaculture systems · Optimization

1 Introduction

Aquaculture provides about half of 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, it exists a strong demand from the public and private sectors of aquaculture research, which have to work towards sustainable production of high quality fish with reduced environmental impact [1]. In the frame of the project AQUAEXCEL (Aquaculture infrastructures for

H. C. Rodrigues et al. (Eds.): *EngOpt 2018 Proceedings of the 6th International Conference on Engineering Optimization*, pp. 1229–1240, 2019.
https://doi.org/10.1007/978-3-319-97773-7_106

excellence in European fish research)[1] we aim to bridge the gap between the scientific community and the fish industry through stimulation of problem-based research and enhanced knowledge transfer.

The improved design of recirculated aquaculture systems (RAS) is needed facing the above mentioned demand for increased fish production as well as increased concern of fish wellbeing, which eventually enhances fish growth [12]. The RAS optimization is a complex problem residing in providing optimum rearing conditions for a specific fish. These optimum conditions encompass different issues, mainly a determined average velocity, e.g., 1–1.5 body length per second for Atlantic salmon [5], low velocity variance assuring homogeneous water quality in entire rearing volume and fast biosolids removal. Last, but definitively not least, the cost of energy and place have to be included into the vector of objective functions to be minimized [3].

Which variables do define the decision space or the set of alternatives of the optimization problem? Apart from the operating conditions and inlet/outlet details, this is the tank geometry which essentially determines the fish culture conditions. Two main geometries have to be considered in the design of aquaculture tanks: rectangular and circular. In general, rectangular tanks are easier to handle and clean than circular tanks, nevertheless, low velocities near corners and poor mixing of water in rectangular tanks lead to the creation of dead volumes, which in turn cause the accumulation of biosolids (faecal solids and uneaten feed) on the tank bottom [8].

An experimental comparison of different geometries and culture conditions would require an enormous effort, hence we focus on developing a CFD (Computational Fluid Dynamics) based methodology helping in the understanding of hydrodynamics of different aquaculture systems. We take various tank geometries and compare their different indices, e.g., average velocity, standard deviation of the velocity magnitude, mean residence time, uniformity index [4], having fixed certain characteristic variable, e.g., the input energy per volume. In this moment, we neither discuss the questions about scaling of a specific cultivation system nor the optimization of operating conditions within a chosen geometry, even less specific geographical conditions or cultivated species. Shortly, we aim to present a tool for solving the problem of suitability of tank hydrodynamics for fish rearing, having in mind that the solution of a related optimization problem depends on how we order or weight different relevant criteria (elements of the vector of objective function).

Our paper is organized as follows: after this introductory section, in Sect. 2, we specify the design parameters and operating conditions for various cultivation systems, we formulate principles for optimal design of fish tanks and discuss some issues concerning fish wellbeing. Here we also present the CFD basics and the most important technical informations about the ANSYS Fluent settings used

---

[1] AQUAEXCEL2020 is a research infrastructure project funded under the EUs Horizon 2020 programme and coordinated by the French National Institute for Agricultural Research (INRA). Its aim is to further support the sustainable growth of the European aquaculture sector, see [1] for more details.

in this study. Section 3 provide the details about CFD simulation process and the resulting characteristic values and indices for each of six configurations. In Sect. 4, the selection of the set of efficient solutions is done and its meaning is discussed. The conclusions, as well as the future goals are presented in final Sect. 5.

2 Materials and Methods

2.1 RAS Geometry

Six different RAS are further examined. The hydrodynamic flow pattern for three basic ‘single’ structures, being rectangular, octagonal and circular, see Fig. 2, and three composed, so-called rectangular multivortex tanks with different baffle arrangement, is investigated using ANSYS CFD (with ANSYS Fluent solver). In order to simplify the subsequent comparison, we chose the same characteristic dimension, inlet/outlet details and operating conditions for each case.

Rectangular Multivortex Tank. Based on the experimental works [8,10], our utmost interest is related to the analysis of the multivortex tanks, see Fig. 1. This experimental device consists of a rectangular tank where at least four rotating flow cells were created, placing the flow inlets tangentially to the cells (with one inlet per cell) and the outlets in the center of each rotating flow cell, see Fig. 1 and [10] and references within there. Such a system combines the advantages of circular tanks (homogeneous distribution of flow velocities) and rectangular tanks (better use of space). However, it remains to study how much is the fluid flow uniform and how it could be controlled. For a tank with given geometry (length L, width W and water depth), the inlet and outlet features are the main factors determining the flow pattern inside the tank. Having the inlet/outlet details fixed, as in the Fig. 1, it seems that there is only the nozzle diameter D_i and the flow rate Q which can be manipulated. Nevertheless, there is another design factor, being a plate baffle, which can be placed into the system in order to enhance the flow pattern within RAS, see Fig. 3.

Tank Operating Conditions and Configurations. Further, in our numerical experiments, we shall test a finite number of configurations and operating conditions defining our feasible set, see e.g. [3]. More precisely, for the six above mentioned configurations we have chosen two operation modes defined either by the same power input P_{in} related to the respective configuration volume, or by the same mean residence time (hydraulic retention time - HRT). These two operation modes were adjusted by the inlet conditions (by the flow rate Q when the inlet diameter D_i was fixed). The outlet is adjusted automatically by a simple hydraulic principle maintaining the same depth in RAS [8,10]. The tank geometry together with initial and boundary conditions determine the flow field within RAS and consequently enabling to calculate certain global (applicable for the whole system) quantities, namely the mean velocity u_{avg}, its standard deviation u_σ and the uniformity index (presented in Tables 1 and 2).

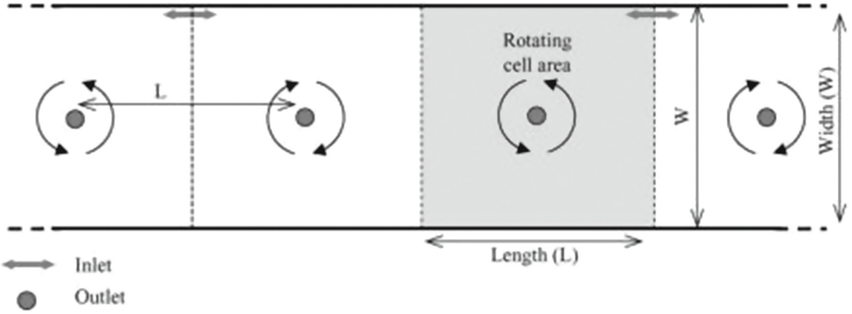

Fig. 1. The detailed geometry of the multivortex tank at UPC, BarcelonaTECH Castelldefels, Spain [8,10].

2.2 Fluid-Dynamic Model of RAS

In the first instance, we model only the incompressible liquid phase (fresh or sea water) in RAS. The simulation of solid particles movement and the influence of fish swimming to the tank hydrodynamics is left to the near future. Therefore, the classical system of Navier-Stokes equations and the continuity equation is used as fluid-dynamic model:

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

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

There are well established methods to solve the above (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 element or finite differences method, 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 both steady and unsteady calculations. In the steady case, it is assumed that time marching proceeds until a steady-state solution is reached.

CFD Simulation and Its Validation

CFD is the art and science of analyzing and simulating systems in which a fluid flow is of central interest and in which heat and mass transfer and chemical reaction may take place. Its advantages over conventional experimental studies are substantial reductions in lead times and development cost, availability to study systems where experiments are not possible and ease of performing a large range of parametric studies for optimization. Although it starts from the design process in industry it has been rapidly used to other application including the agro-environmental applications [5–7]. The main goal of the present paper is to present an application of CFD for RAS design and eventually RAS optimization. Although we got quiet satisfactory results, comparing with the experimental works [8–10], there is a lot of challenges for CFD modelers in analyzing RAS hydrodynamics using CFD, e.g., the description of influence of the fish swimming on the flow pattern [5,11] and the quantification of fish preferences [12].

The very important issue is the validation of CFD model, which 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 [2]. Validation should depend on direct comparison between CFD computed result and measured experimental result. This issue will be treated only broadly in Sect. 5.

2.3 Design and Operational Variables of RAS and the Efficient Solutions of Multicriteria Optimization

Multicriteria (multi-objective, vector) optimization conversely to the single objective function look for the set of efficient solutions rather than one optimum solution. This subsection prepare some key notion related to the multicriteria optimization on a feasible set, or the set of alternatives of the decision (design) problem related to both design and operational variables (parameters, factors) of a general RAS. Obviously, the most complex and important task is related with an appropriate setting of the set of *objective functions* (or criteria of efficiency) $f_i, i = 1, \cdots, m$. These criteria usually encompass the rearing conditions for fish (e.g., a determined average velocity, low velocity variance) and quantify the mean time of biosolids removal and the cost of energy and place. The scalarization of these partial criteria into a single criterion is supposed to be provided by communication with experts in the field of RAS.

The variables needed for the partial criteria evaluation are proportioned, in our case, by the process model in form of (1), announced in the previous section. This fluid-dynamic model provides a coherent description of the entire tank hydrodynamics on the level of RAS operation, including the operating mode, and depend on a set of parameters forming the decision space (the feasible set is a subset of the decision space). As follows, we formulate the following terms needed for the statement of an optimization problem, denoted by $(\mathcal{P})$:

- **Design variables:** Let us supposed the RAS structure is parameterized, i.e. the RAS geometry is determined by p time independent design variables $d = (d_1, ..., d_i)$, $i = 1, ..., p$. These variables d must be nonnegative and bounded by certain maximal admissible value, or they must lie in an admissible set: $d \in \mathcal{U}_{ad}^d$.
- **Operational variables:** Let us supposed the RAS is controlled by means of the operational (generally time dependent) control variables $g(t)$, e.g., the flow rate Q, which determines the intensity of mixing, the inflow rate of oxygenated water and the mean residence time HRT. For the technological reasons, some constraints are applied (e.g. non-negativity and boundedness): $g \in \mathcal{U}_{ad}^c$.

- **State variables:** The state variables $c = (c_i(x,t))$, $i = 1, ..., n$, are describing the state of the system, in our case it is the flow pattern and eventually concentrations of some conservative quantities as dissolved oxygen or solid particles.[2] There could be some thresholds or constraints, e.g. specified by the relation $c \in \mathcal{U}_{ad}^s$.
- **Statement of the problem of multicriterial optimization:** Let us suppose, we have multiple criteria of efficiency, the vector of partial criteria $f = (f_1, ..., f_m)$ and the objective functional J (to be minimize) as a result of convolution (scalarization) of the partial criteria f_i, $i = 1, ..., m$, into a single criterion. It exists a set of restrictions and certain functions of the state variables (depending on design and operational variables) involved in the objective functional.

Then the optimal control & design problem $(\mathcal{P})$ consists of:

$$(\mathcal{P})\, minimize\, J\,(c(d,g))\;\; such\, that\, d\, \in \mathcal{U}_{ad}^d,\; g\, \in \mathcal{U}_{ad}^c\, and\, c\, \in \mathcal{U}_{ad}^s\, verifies\,(1). \tag{2}$$

Remark 1: To demonstrate that the problem $(\mathcal{P})$ admits, at least, a solution, and to derive optimality conditions is out of the scope of this paper.

3 Results

A two-dimensional geometry representing a simplified version of various geometrical configurations was created in ANSYS Fluent [4], see Figs. 2, 3 and 4. For the single tank configurations (square, octagonal, circular), a mesh with around 18 thousand elements was used. For the multivortex tank configurations, a mesh with around 65 thousand elements was used in our simulations. Refined mesh elements near walls were created so that dimensionless thickness Y^+ was around 1 to properly capture viscous sublayer in the turbulent flow regime. SST $k-\omega$ turbulence model, boundary conditions velocity-inlet and pressure-outlet, and steady-state case were used in our CFD model [4].

The power input (unit: J/s) of the inlet stream can be expressed as multiple of kinetic energy density (unit: J/m^3) and volumetric flow rate (unit: m^3/s)

$$P_{in} = \frac{1}{2}\varrho u_{in}^2\, Q\ , \tag{3}$$

where u_{in} represents the inlet velocity, ϱ liquid density and Q is volumetric flow rate. This energy can be then related to the system volume V

$$\varepsilon_V = \frac{P_{in}}{V} = \frac{1}{2}\varrho u_{in}^2\, \frac{Q}{V}\ . \tag{4}$$

---

[2] Special attention should be paid to the temperature. In order to model the non-isothermic processes, we would add the heat transport equation to our PDE (1) describing the tank hydrodynamics as well.

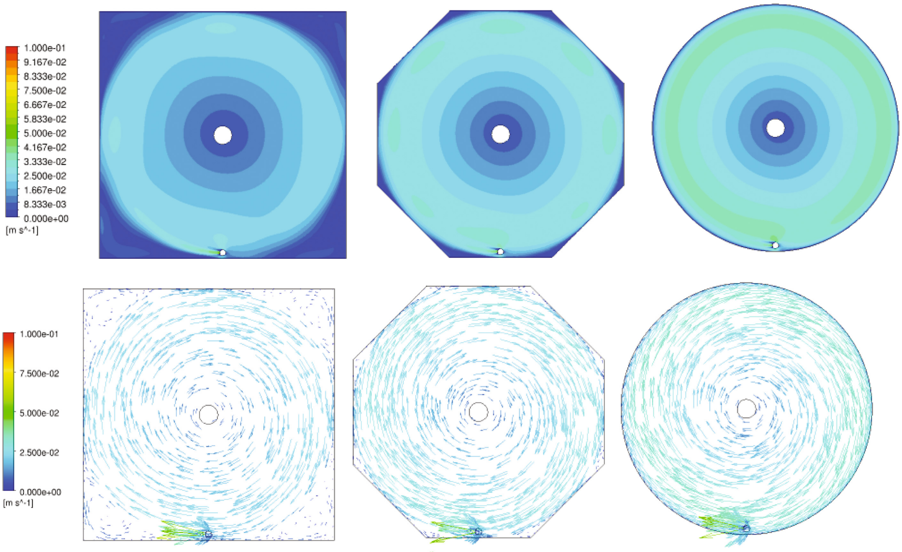

Fig. 2. Comparison of velocity contours (upper part) and velocity vectors (lower part) for the single-tank geometry configurations (from left to right: square, octagonal, and circular). The color (and length of the arrows in lower part) is representing the velocity magnitude.

Substituting for the inlet volumetric flow rate $Q = u_{in} A_{in}$, where A_{in} represents area of inlets ($A_{in} = 0.010276\,\mathrm{m}^2$ for single inlet in our simplified 2-D geometry), we get relation between the inlet velocity and power input as

$$u_{in}^3 = \frac{2V\varepsilon_V}{\varrho A_{in}} \; . \tag{5}$$

The above relation simply computes the inlet velocity, one of the boundary conditions for the PDE (1), when the power input and the inlet geometry details are specified.

One of the most important objective functions contains some measure of the uniformity of flow pattern. ANSYS Fluent has the uniformity index γ_a embedded as follows

$$\gamma_a = 1 - \frac{\sum_i |u_i - u_{avg}| A_i}{2|u_{avg}| \sum_i A_i} \; . \tag{6}$$

γ_a represents the variability of the velocity over the surface ($\sum_i A_i$ is the surface of all mesh elements in our 2-D geometry), when the value $\gamma_a = 1$ means the highest uniformity. u_{avg} represents the area-weighted average of the velocity

$$u_{avg} = \frac{\sum_i u_i A_i}{\sum_i A_i} \; . \tag{7}$$

Another way how to quantify the uniformity (or the non-uniformity) of the velocity flow field is using the standard deviation u_σ. It is computed using the mathematical expression below

$$u_\sigma = \sqrt{\frac{\sum_i (u_i - u_{avg})^2}{n}}\,, \tag{8}$$

i.e., it is not weighted by the individual mesh element size A_i as the γ_a index.

In the following tables we present the summary of CFD results for six different geometrical configurations with constant mean residence time $\bar{t} = 1600\,\text{s}$ (Table 1) and with constant power input (Table 2), respectively.

Table 1. Summary of CFD results for various geometrical configurations with constant mean residence time $\bar{t} = 1600\,\text{s}$.

	u_{in} [m/s]	u_σ [m/s]	γ_a [−]	u_{avg} [m/s]	V [m^3]	Q [m^3/s]	$\bar{t}$ [s]	ε_V [W/m^3]
Multi, baffles	0.0604	9.192e-03	0.7921	0.0160	3.976	2.48e-03	1600	1.140e-03
Multi, triangle	0.0599	8.883e-03	0.8100	0.0162	3.937	2.46e-03	1600	1.118e-03
Multi, no baffles	0.0605	8.934e-03	0.8054	0.0162	3.982	2.49e-03	1600	1.143e-03
Single, square	0.0605	9.463e-03	0.7496	0.0153	0.995	6.22e-04	1600	1.143e-03
Single, octagonal	0.0501	9.031e-03	0.8219	0.0186	0.824	5.15e-04	1600	7.826e-04
Single, circular	0.0475	8.988e-03	0.8534	0.0220	0.780	4.88e-04	1600	7.029e-04

Table 2. Summary of CFD results for various geometrical configurations with constant power input $\varepsilon_V = 1 \times 10^{-3}\,\text{W/m}^3$.

	u_{in} [m/s]	u_σ [m/s]	γ_a [−]	u_{avg} [m/s]	V [m^3]	Q [m^3/s]	$\bar{t}$ [s]	ε_V [W/m^3]
Multi, baffles	0.0579	8.772e-03	0.7915	0.0152	3.976	2.38e-03	1671	1.000e-03
Multi, triangle	0.0577	8.504e-03	0.8095	0.0156	3.937	2.37e-03	1660	1.000e-03
Multi, no baffles	0.0579	8.441e-03	0.8045	0.0152	3.982	2.38e-03	1673	1.000e-03
Single, square	0.0579	9.039e-03	0.7489	0.0145	0.995	5.95e-04	1673	1.000e-03
Single, octagonal	0.0544	9.773e-03	0.8233	0.0203	0.824	5.59e-04	1474	1.000e-03
Single, circular	0.0534	1.004e-03	0.8558	0.0250	0.780	5.49e-04	1423	1.000e-03

The simplified 2-D geometry was selected in order to have low computational requirements in this initial CFD study. Simulation of one geometrical case took approximately from 5 to 20 min on Intel CPU i7-2600 3.40GHz. If we wanted to switch to a full 3-D geometry, the computational demands would, of course, substantially increase. Having similar mesh resolutions, the number of mesh elements would increase from tens of thousands in 2-D to several millions in a 3-D variant. The computational time is more or less proportional to the number of mesh elements, and it could be decreased by parallel processing on multiple processors (cores). But in most cases, the communication between parallel processes represents an overhead which can more or less decrease the efficiency of parallel computations. For example, using 10 parallel processes, we could speed-up the simulations by factor 5 only. Therefore, with a full 3-D geometry, we should expect the simulation time of a single case in order of 10 or more hours.

4 Discussion

A numerical study of RAS hydrodynamics based on a CFD code was completed. Six different geometrical configurations for the same inlet/outlet design and various operational conditions (flow rate Q) were tested. We proposed and effectuated the calculations for two operation modes: first, with mean residence time set to some reasonable value (1600 s), second, with constant power input.[3] We expected, based on the intuition and the experimental measurements published in [10], that there will be substantial differences between each of six configurations. Namely between multivortex systems with and without baffles, because a plate baffle with an appropriate position and size shall cause higher symmetry and uniformities (lower velocity variance) than do the same RAS without baffle. Indeed, for both operation modes, the multivortex tanks with (and without!) baffles placed between two water inlets exhibit higher uniformities than do single square cells, see Tables 1 and 2. Nevertheless, looking at Figs. 3 or 4 and the quantities presented in both tables, we encounter only small differences among

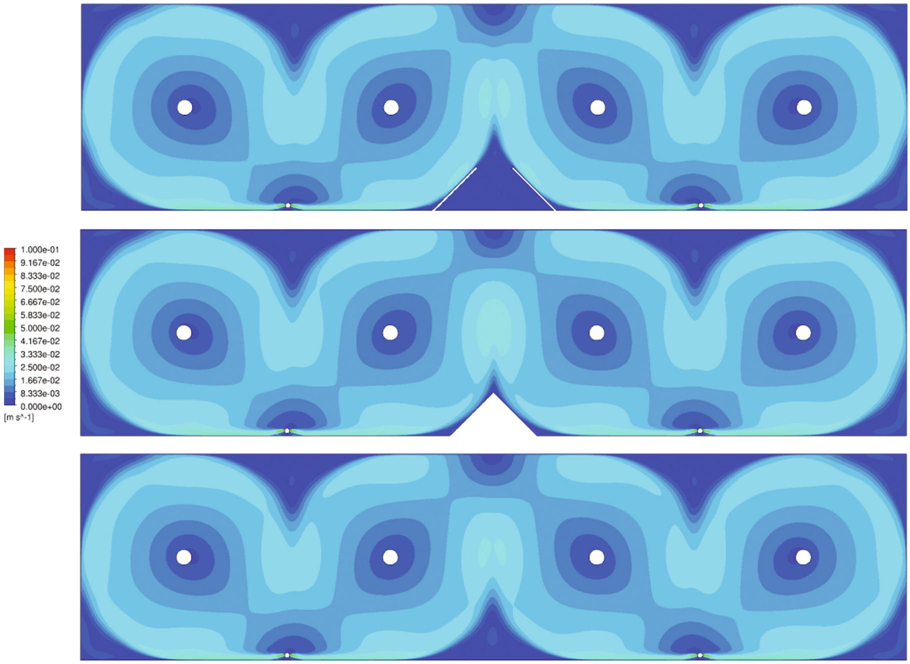

Fig. 3. Comparison of velocity contours for the multivortex geometry with baffles (top), triangle-shaped baffle (middle), and without baffles (bottom).

---

[3] The constant power input per volume, simultaneously with the constant cross section A_{in}, means that the impulse force linked with one outlet introduced in [8] is the same for all six configurations.

respective multivortex tanks, see corresponding values in the three top rows of Tables 1 and 2 (depicted as multi-baffles, multi-triangle, multi-no baffles).

Finally, taking the uniformity index as the only criterion for the RAS selection, we claim that the best geometrical configuration is the circular one, followed by the octagonal, which can be seen as the single rectangular cell with four baffles. Clear looser is the single square RAS. Concerning *multivortex tanks*: according to our 2-D calculations and the operating conditions (not entirely realistic), we state that the mean velocities, their standard deviations and the uniformity indices for all three multivortex tanks do not differ substantially (considering the numerical error bounds). This very last result induce us to perform new full 3-D CFD simulations of multivortex RAS in order to figure out where is the cause of this unexpected behavior.

Nevertheless, once having a reliable method to calculate the fluid flow within the RAS, we can select the optimum RAS geometry and subsequently we can address two other important features in RAS design & control: (i) to ensure the optimal rearing conditions including the fish exercising, and (ii) to maximize the capability of the system for self-cleaning.

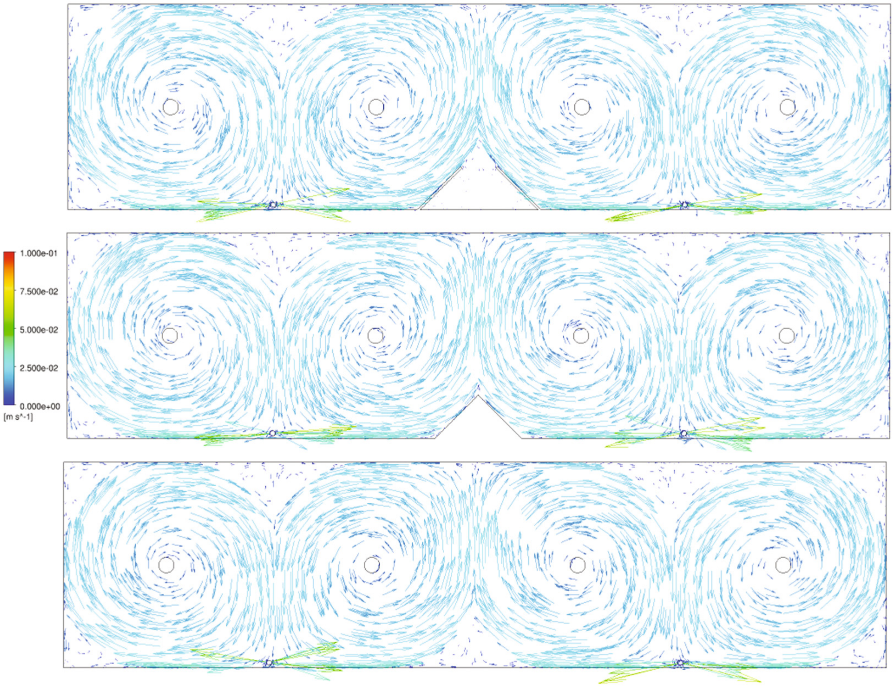

Fig. 4. Comparison of velocity vectors for the multivortex geometry with baffles (top), triangle-shaped baffle (middle), and without baffles (bottom).

5 Conclusions

Our work represents a step towards the use of CFD for aquaculture tank design and control. Despite the fact that CFD calculations may not be sufficiently exact, the degree of error is usually within reasonable bounds for engineering purposes. Thus, CFD analysis is able to become a standard tool for RAS design and optimization reducing cost of research and engineering development. It is clear that intending to perform RAS optimization experimentally, on a real physical model, would require an enormous effort. Here, we have shown how the CFD code ANSYS Fluent can use the (embedded) mathematical model and perform the experiments *in silico*. Although we did neither propose nor apply a real optimization procedure, the best variant can be chosen 'manually' by a simple selection between the finite number of variants.

Outgoing work can focus both on engineering and computational sides; first, we aim to determine (and then implement into a CFD code) the influence of fish swimming on fluid flow pattern inside RAS, and second, a realistic vector of objective functions should be set up and tested within an optimization procedure.

Acknowledgements. 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), and by the long-term strategic development financing of the CTU.

References

1. AQUAculture infrastructures for EXCELlence in European fish research towards (2020). http://www.aquaexcel.eu/
2. AIAA, Guide for Verification and Validation of Computational Fluid Dynamics Simulation. AIAA-G-77. American Institute of Aeronautics Astronautics (1998)
3. Ehrgott, M.: Multicriteria Optimization. Springer, Berlin (2000)
4. ANSYS Fluent. http://www.ansys.com/Products/Fluids/ANSYS-Fluent
5. Gorle, J.M.R., Terjesen, B.F., Mota, V.C., Summerfelt, S.: Watervelocity in commercial RAS culture tanks for Atlantic salmon smoltproduction. Aquacult. Eng. **81**, 89–100 (2018). https://doi.org/10.1016/j.aquaeng.2018.03.001
6. Hyo, M.J., Han, S.Ch., Se, H.K., Seuk, Ch.Ch., Kang Y.B.: Fluid flow characteristics in the aquaculture tank for a breeding fish. KSME Int. J. **18**(12) , 2265–2272 (2004). https://doi.org/10.1007/BF02990231
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. Comput. Electron. Agric. **93**, 168–183 (2013). https://doi.org/10.1016/j.compag.2012.09.006
8. Oca, J., Masaló, I.: Design criteria for rotating flow cells in rectangular aquaculture tanks. Aquacult. Eng. **36**, 36–44 (2007)
9. Oca, J., Masaló, I.: Flow pattern in aquaculture circular tanks: influence of flow rate, water depth, and water inlet & outlet features. Aquacult. Eng. **52**, 65–72 (2013)
10. Masaló, I., Oca, J.: Hydrodynamics in a multivortex aquaculture tank: effect of baffles and water inlet characteristics. Aquacult. Eng. **58**, 65–72 (2014)