Numerical Modelling of the Interaction between a Fish Net and Fluid using CFD

Tobias Martin*, Arun Kamath and Hans Bihs

* Department of Civil and Environmental Engineering
Norwegian University of Science and Technology
Høgskoleringen 7a, 7491 Trondheim, Norway
e-mail: tobias.martin@ntnu.no

ABSTRACT

The aquaculture industry has seen a strong growth over the last decades due to its potential to meet the rising global food demand. Offshore fish production becomes relevant as the size of the structures increases and greater concerns about the environmental impact on the nearshore zone arise. Here, severe environmental loadings from high energy sea states necessitate accurate analysis and calculation for the design of reliable and economical marine fish farm structures. In the past, separated studies on these aspects have been conducted using experimental or simple numerical tools. The latter approach might, however, be misleading due to the strong interrelationship between the single components of the structure. In contrast, computational fluid dynamics can be used to understand the structural and environmental challenges in the operation of the whole structures by studying the forces on and the fluid dynamics in and around the cages.

Recently, a numerical model for the determination of the deformed shape of nets with elastic twines was developed [1]. It is a quasi-static approach which leads to an efficient overall computation due to missing time step restrictions. In this paper, the current efforts of coupling this model to a solver for the Navier-Stokes equations are presented. The main focus of the research lies on the projection of the fluid forces on the net in order to incorporate hydrodynamic loads and modelling of the net's influence on the fluid using a porous media approach.

The applied CFD model is the open-source solver REEF3D. The solver has been widely applied to study various problems in the field of wave hydrodynamics and fluid-structure interaction. The incompressible Reynolds-averaged Navier-Stokes equations are solved for water and air using a one-fluid approach. A level set method with an equation-based reinitialisation algorithm is implemented for capturing the free surface implicitly. For the discretisation, a finite difference method on a staggered structured grid is employed. Convection terms are discretised using a fifth-order accurate WENO scheme, and the equations are solved in time using the third-order accurate TVD Runge-Kutta method. An immersed boundary method based on the directional ghost cell method represents the interaction of fluid and floating structures [2].

REFERENCES

[1] Martin, T., Schacht, S., Riesen, P. and Paschen, M., Efficient implementation of a numerical model for flexible net structure. *Ocean Engineering*, Vol. **150**, pp. 272–279, (2018).

[2] Bihs, H. and Kamath, A., A combined level set/ghost cell immersed boundary representation for floating body simulations. *Int. J. Numer. Meth. Fluids*, Vol. **83**, pp. 905–916, (2017).