Ocean Engineering 142 (2017) 597–615

Contents lists available at ScienceDirect

Ocean Engineering

journal homepage: www.elsevier.com/locate/oceaneng

Development of a numerical model for fluid-structure interaction analysis of flow through and around an aquaculture net cage

Hao Chen*, Erik Damgaard Christensen

Section of Fluid Mechanics, Coastal and Maritime Engineering, Department of Mechanical Engineering, Technical University of Denmark, DK-2800 Kgs. Lyngby, Denmark

ARTICLE INFO

Keywords:
Porous media model
Lumped mass model
Fluid-structure interaction analysis
Aquaculture net cage
Coupling scheme

ABSTRACT

In the present work, we developed a numerical model for fluid-structure interaction analysis of flow through and around an aquaculture net cage. The numerical model is based on the coupling between the porous media model and the lumped mass structural model. A novel interface was implemented to ensure efficient data exchange and element mapping between the fluid and structural solver via random-access memory. The main idea is to apply a static mesh in the fluid model, in case that large deformation of the net structure reduces the quality of the mesh. Then the geometry of the net cage was approximated by a set of dynamic porous zones, where the grid cells were updated at every iteration based on the transferred nodal positions from the structural model. A time stepping procedure was introduced, so the solver is applicable in both steady and unsteady conditions. In order to reduce the computational effort, sub-cycling was applied for the structural solver within each time step, based on the quasi-steady state assumption. The numerical model was validated against experiments in both steady and unsteady conditions. In general, the agreement is satisfactory.

1. Introduction

Aquaculture has been an important resource for food production in the world, and globally it is in a phase of steady expansion. As fresh water aquaculture has been increasingly constrained, space and water availability is driving aquaculture growth towards mariculture, from the bays and fjords with sheltered water to more exposed sites with large currents and waves. Therefore, the design for future offshore fish cages requires more accurate analysis and calculations.

Different numerical models have been proposed for analysis of hydrodynamic forces on the net structures. In general, they can be categorized into two approaches. One is the Morison type force model, where the net was modeled as individual twines. In many cases due to a large number of mesh in a net cage, the physical net cage was represented by an equivalent system of mesh with less twines. Morison equation was applied to calculate the forces on each twine, see e.g. Li et al. (2006), Moe et al. (2010), Tsukrov et al. (2002). Meanwhile, screen type force models have also been developed, where the net was divided into a number of screen elements. Each screen element is subject to the same forces as on the twines and knots that are being represented. Due to a deflection of the flow through the screen, the resulted total force on a screen is not in the inflow direction, and it is usually decomposed into a drag and a lift component. Løland (1993) derived the drag and lift force coefficient based on fitting of the data from laboratory tests in Rudi et al. (1988). In Kristiansen and Faltinsen (2012), instead of using the curve fitting method, the drag and lift force coefficient were obtained as a function of solidity ratio, the angle between the panel normal and the flow direction, and the drag force coefficient for the twines of the net in steady current, which is Reynolds number dependent. Kristiansen and Faltinsen (2015) demonstrates the application of this force model in waves. The elastic floating ring was also taken into account in the numerical model. Therefore, a complete rational model for a floating fish cage was set up.

Recently, the computational fluid dynamic (CFD) method combined with the porous media model was applied for simulating flow through such kind of porous structures in e.g. Patursson et al. (2010) and Zhao et al. (2013a, 2013b) etc. The net structure was modeled as a layer of porous media. An extra resistance was added in Navier-Stokes equations to represent the effects of the net structure on the fluid. The advantage of this method is that it is not necessary to model the detailed geometry of the net structure, which keeps the computational time on a reasonable level. Furthermore, both the hydrodynamic force on the net cage and the flow field through and around the net cage could be simulated in the numerical model. From the validation in the above-mentioned works, the numerical results agree well with the laboratory tests.

* Corresponding author.
E-mail address: hchen@mek.dtu.dk (H. Chen).

http://dx.doi.org/10.1016/j.oceaneng.2017.07.033
Received 31 October 2016; Received in revised form 6 June 2017; Accepted 10 July 2017

However, in reality, net structures are quite deformable under current and waves. The deformed structure will in turn affect the flow field. So, this is a typical fluid-structure interaction (FSI) problem. One needs to solve the governing equations in both the fluid and solid domain. In general, the numerical procedure to solve FSI problems could be broadly classified into two approaches: monolithic approach and partitioned approach (Hou et al., 2012). The monolithic approach treats the fluid and solid problems in the same mathematical framework. Hereby a single system of equations is formed finally for the entire problem in both the fluid and solid domain, which is solved at each time step. This indicates that the coupling is on a matrix level, and is fundamentally implicit, see e.g. Hübner et al. (2004), Le Tallec and Mouro (2001) and Michler et al. (2004) etc. This approach can potentially achieve better stability and accuracy, but it requires substantially much more resources and expertise to develop and maintain such a specialized code (Hou et al., 2012). Hereby instead of developing a monolithic solver from scratch, we decided to apply the partitioned approach. This means that the fluid and structural problem are solved separately with their respective numerical solver and mesh discretization. In the present work, the available CFD solver combined with a porous media model in the open source toolbox OpenFOAM is utilized as the fluid solver, while a custom lumped mass solver for the net cage is implemented as the structural solver.

Comparing with a conventional FSI solver, the coupling method between these two solvers is different for this particular problem, owing to its distinct characteristics. Usually a conventional FSI problem is a surface coupling problem, where the solid domain and the fluid domain are connected but not overlapped. There exists a clear interface which separates the fluid and solid domain, namely the boundary of the solid body. The interfacial conditions are explicit where the kinematic and dynamic boundary condition need to be fulfilled on the interface. However, for this problem, the geometry of the solid body (namely the net structure) is not resolved in the fluid solver. Instead, its effects are taken into account in a fictitious manner via an extra forcing term, i.e. the resistance in the Navier-Stokes equations. The forcing term is only active in the porous zone occupied by the net structure, and it is switched off anywhere else. Therefore, in the numerical model, the interface to separate the fluid and solid domain is not exactly reproduced, and the fluid is continuous throughout the whole computational domain. The deformation of the net structure is not represented by deformation of the interface, but motion of the resistance in the porous zones.

In principle, motion of the resistance could be realized in two ways in the numerical model. One is via motion of the internal subset of the mesh which represents the porous zones. This requires applying deforming mesh techniques. The internal subset of the mesh should conform with the deformation of the net structure, while the complementary part of the mesh is deformed based on the motion of the internal subset of the mesh. The benefit of this method is that the net deformation can be described accurately, since the motion of the internal subset of the mesh can in general match the deformation of the net very well. However, by using this method, the computational mesh may be significantly distorted, considering the negligible bending and compression stiffness of the netting materials. Therefore, we choose the second method, where a static mesh is applied. The instantaneous deformation of the net structure is represented by updating the grid cells in the porous zones, based on the solution from the structural solver. Conceptually this is similar to the immersed boundary method (Mittal and Iaccarino, 2005) for moving objects. The key point of this method is to find a fast and reliable algorithm for searching the grid cells that belong to the updated porous zones at each time step. Fig. 1 gives an example on how these two methods are applied to represent a rotating porous zone. It is observed that by using the deforming mesh technique, the geometry of the rotating porous zone is accurately reproduced. However, this leads to a distortion in the complementary part of the mesh. It should be mentioned that Fig. 1(b) is only a graphic illustration on the deforming mesh technique, rather than any exact solution of a specific deforming mesh method. Actually, there exists plenty of methods, but essentially the nature of which is to solve different equations to relocate the grid cells for the updated moving boundary at every time step. Depending on the method that is applied, the distortion of the computational mesh can be either intensified or attenuated. But in general, for large amplitude of motion, especially for rotation motion, the distortion is significant irrespective of the method that is applied. However, in Fig. 1(a) with a static mesh, this issue is resolved. But we notice that the real boundary of the porous zone has zigzag fashion. Hereby a relatively fine mesh is required in the vicinity of the moving porous zones.

To the authors' knowledge, among those previous works on CFD simulation of flow though and around net structures, few considered the effects of net deformation, and they all applied the second method with static mesh. In Devilliers et al. (2016), a special solver was implemented for FSI analysis of current flow through net structures. The fluid solver solved pseudo-compressible Navier-Stokes equations, and the structure code *DynamiT* was applied to predict the net deformation, which approximated the net as a set of rigid bars. The resistance source term was estimated by Landweber-Ritchmeyer mechanic hypothesis, and the coupling was via output files. Advanced adaptive mesh refinement technique was applied to increase the resolution of the mesh in the net area. In Bi et al. (2014a, 2014b), attempts have also been made to couple the porous media model with a lumped mass structural model. The coupling was based on the concept of "iteration". Under each iteration, steady state condition was assumed. The numerical model reached convergence very fast, usually under 1 – 3 iterations. In the present paper, we will further improve the solver presented in Bi et al. (2014a, 2014b). The main idea behind it is to achieve the FSI analysis in both steady and unsteady conditions. Hereby the linear and nonlinear waves can be properly generated in the numerical model.

The present paper is organized as follows. The fluid and structural solver that are applied in the numerical model are described in Section 2. Then Section 3 gives a thorough description on the coupling scheme between these two solvers. In Section 4 numerical study is carried out using the present model for three sets of cases, followed by a summary and conclusion in Section 5.

2. Description of the fluid and structural solvers

This is a typical hydro-elastic problem where the governing equations need to be solved in both fluid and solid domain. In the present work, we adopted an existing solver in OpenFOAM as the fluid solver, and implemented a lumped mass solver for the structural deformation of the net. Below in this section, these two solvers are described in detail.

2.1. CFD solver

The net structure in the fluid domain was represented by a sheet of porous media. This requires that the fluid solver has the capability on analysis of flow through porous structures. In Jensen et al. (2014) the governing equations on porous media flow were revised. The relevant library in OpenFOAM was re-implemented based on the new formulation, and it was released as open source together with the *waves2Foam* toolbox developed in Jacobsen et al. (2012). In the present work, this library was applied for both single-phase and two-phase flow solver in OpenFOAM. Below a brief description is given on it.

2.1.1. Governing equations

The governing equations for the CFD solver are the volume averaged Reynolds averaged Navier-Stokes (VARANS) equations:

$$\nabla\cdot\langle\overline{\boldsymbol{u}}\rangle = 0 \tag{1}$$