Available online at https://link.springer.com/journal/42241
http://www.jhydrodynamics.com
Journal of Hydrodynamics, 2020, 32(2): 212-233
https://doi.org/10.1007/s42241-020-0021-5

Numerical techniques for coupling hydrodynamic problems in ship and ocean engineering *

Xiao-song Zhang, Jian-hua Wang, De-cheng Wan
Computational Marine Hydrodynamics Lab (CMHL), State Key Laboratory of Ocean Engineering, School of Naval Architecture, Ocean and Civil Engineering, Shanghai Jiao Tong University, Shanghai 200240, China

(Received December 29, 2019, Revised January 18, 2020, Accepted January 20, 2020, Published online April 23, 2020)

Abstract: Most hydrodynamic problems in ship and ocean engineering are complex and highly coupled. Under the trend of intelligent and digital design for ships and ocean engineering structures, comprehensive performance evaluation and optimization are of vital importance during design. In this process, various coupling effects need to be accurately predicted. With the significant progress of computational fluid dynamics (CFD), many advanced numerical models were proposed to simulate the complex coupling hydrodynamic problems in ship and ocean engineering field. In this paper, five key coupling hydrodynamic problems are introduced, which are hull-propeller-rudder coupling, wave-floating structure coupling, aerodynamic-hydrodynamic coupling, fluid structure coupling and fluid-noise coupling, respectively. The paper focuses on the numerical simulation techniques corresponding to each coupling problem, including the theories and the applications. Future directions and conclusions are provided finally.

Key words: Coupling hydrodynamic problems, numerical techniques, ship and ocean engineering

Corresponding authors' biography

Prof. De-cheng Wan is distinguished professor of Shanghai Jiao Tong University (SJTU), chair professor of Chang Jiang Scholar, distinguished professor of Shanghai Eastern Scholar, Shanghai Excellent Academic Leader. He is awarded the most cited researchers in 2018 by Elsevier, and has delivered over 80 invited or keynote presentations in international conferences. Currently, Prof. Wan is Director of Computational Marine Hydrodynamics Lab (CMHL, https://dcwan.sjtu.edu.cn/) at SJTU, member of ISOPE Board of Directors, Chair of ISOPE International Hydrodynamic Committee, member of advisor committee of International Towing Tank Conference (ITTC), chair of international scientific committee of OpenFOAM workshop, standing council member of Association of Global Chinese Computational Mechanics, Associate Editor-in-Chief of Journal of Hydrodynamics, and member of editorial board of Ocean Engineering and Applied Ocean Research. His research interest is mainly on computational marine and coastal hydrodynamics, numerical marine basin, nonlinear wave theory, wave loads on structures, numerical analysis of riser vortex-induced vibration (VIV) and platform vortex-induced motion (VIM), fluid-structure interaction, offshore wind turbine and other offshore renewable resources, etc. In these areas, he has published over 500 papers and carried out more than 50 projects on marine computational hydrodynamics. His remarkable works of development of numerical solvers in ship and ocean engineering have been recognized by the world-wide researchers in the field of marine hydrodynamics.

Introduction

Complex coupling is an important characteristic of ship and ocean engineering problems. Marine engineering equipment often consists of complex multi-system. For example, ship navigation is dependent on the cooperation of hull, propeller and rudder. The motion and force of different systems interact with each other and the quality of any system will affect the safe operation of the whole system. In addition, the operational environment of ship and

* Project supported by the National Natural Science Foundation of China (Grant No. 51879159, 51809169 and 51909160), the National Key Research and Development Program of China (Grant Nos. 2019YFB1704200, 2019YFC0312400).
Biography: Xiao-song Zhang (1996-), Male, Ph. D. Candidate, E-mail: xiaosong_zhang@sjtu.edu.cn
Corresponding author: De-cheng Wan,
E-mail: dcwan@sjtu.edu.cn

marine engineering equipment involves the coupling of multi-physics fields. The most fundamental is the coupling of complex waves and floating structures. Besides there is also a coupling problem of aerodynamics and hydrodynamics, which plays an important role in the work of floating wind turbine. Besides, the hydrodynamic, structural and acoustic characteristics of the equipment have significant mutual interference and coupling characteristics. Modern ship and marine engineering structure design is oriented to comprehensive performance, which requires considering the above coupling problems in performance evaluation.

Over the past 30 years, computational fluid dynamics (CFD) has advanced by leaps and bounds. CFD simulation has been widely applied to the research of various ship and ocean engineering problems. Simulation-based design (SBD) approach is replacing the now old-fashioned build-and-test approach so that model testing is only required at the final design stage[1]. Numerical simulation of coupling problem is one of the major challenges in research. Many numerical algorithms are proposed in order to better predict the comprehensive performance. The numerical simulation techniques based on these numerical algorithms have further promoted CFD to become an important design method for ship and Marine engineering structures. This paper focuses on the principle of these numerical simulation techniques and their applications on ship and ocean engineering field.

In this paper, typical ship and ocean engineering coupling problems are classified into five categories: ship hull-propeller-rudder coupling, wave-floating structure coupling, aerodynamic-hydrodynamic coupling, near-far field coupling and fluid-structure coupling. Each problem is introduced in a separate section and the corresponding numerical simulation techniques are described in the section. A brief conclusion is drawn in the final section and the future work is discussed as well.

1. Hull-propeller-rudder coupling

The mutual cooperation between ship hull, propeller and rudder is one of the most complex multi-system ship hydrodynamic problems. Each of these three components has independently 6DOF motion, and they affect each other at the same time. Numerical simulation of hull-propeller-rudder coupling can be divided into two methods, one is body force method independent on propeller geometry, and the other one is directly modelling the propeller geometry. In addition, the second method can be further subdivided into sliding grid method and overset grid method. Numerical studies using the above three methods will be introduced in the following sections.

1.1 *Body force method*

By using body force method to simulate propeller, the complex geometry modelling and mesh generation for propeller can be avoided, leading to a significant increase in computational efficiency. In the body force method, a source term whose action region equals to the propeller disk is added to the momentum equation to represent the effect of propeller. There are three main calculation algorithms for body force: blade element momentum theory, equivalent thrust disk model and lifting-surface/line model. The body force method has been widely used in the early stage to simulate the coupling of hull, propeller and rudder due to its low computational cost.

Stern et al.[2] performed ship self-propulsion simulation in 1988, the interaction between ship hull and propeller was modelled by body force based on lifting surface method. The same method was adopted by Kawamura et al.[3] to investigate the flow around ship with propeller. Choi et al.[4-5] developed a propeller body force code based on vortex lattice method and combined it with commercial CFD software FLUENT. With the help of this software, they calculated the coupling of hull and propeller for various ships. Phillips et al.[6] studied the difference of three different body force models in simulating the interaction between the propeller and rudder. The first is the equivalent thrust disk model, because the torque of the propeller is ignored in this model, the prediction accuracy of the pressure on the rudder is unsatisfactory. The second is the Hough and Ordway model, the rotational effect of the propeller is considered in this model, thus the pressure on the rudder is well predicted. However, the reaction of the rudder to the propeller cannot be calculated. Finally there is the blade element momentum theory, by the combination of this method and RANS model, the interaction between propeller and rudder can be well simulated.

The body force method provides a convenient way for the numerical simulation of complex maneuvering motion of ships. Simonsen and Stern[7] carried out RANS simulation for the maneuvering motion of oil tanker Esso Osaka. The coupling of hull-propeller-rudder is solved by body force model based on potential flow method. Calculated results are basically in agreement with the measured results. Traditional body force model was further improved by Dubbioso et al.[8-9] to account more accurately for the propeller loading. They adopted an in-house CFD solver χnav to simulate Zig-zag maneuvering motion of a tanker with double rudders. Velocity in the propeller disk modelled by body force method and nearby streamlines can be seen in Fig. 1. The first transcendence angle was in good agreement with the experimental value, but the second simulated result was larger, which indicates that the modified propeller body force model still needs to be further improved. Mofidi et al.[10] developed a coupled CFD/body force propeller code and simulate manoeuvres of ships. The influence of the rudder in the wake of a propeller on the two components was investigated. Flow separation on the rudder was not well captured, since the propeller turbulence reaching the rudder is underestimated in the coupled approach. Kaidi et al.[11] used body force to simulate propeller and studied a complex bank-propeller-hull coupling problem. The effect of ship-bank distance, ship speed and the advance ratio of the propeller were analysed in detail.

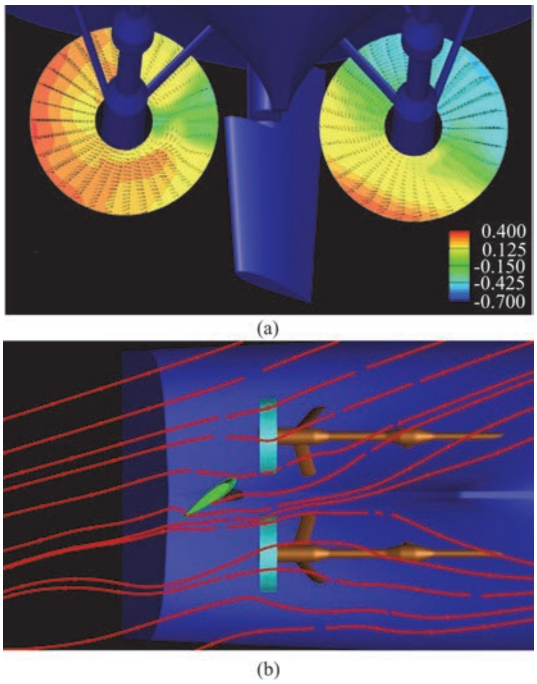

Fig. 1 (Color online) Velocity in the propeller disk modelled by body force method and nearby streamlines[9]

1.2 *Sliding grid method*

Compared with body force method, modelling the propeller geometry significantly improves the accuracy of propeller flow simulation in hull-propeller-rudder coupling. With the development of high performance computing equipment and numerical methods, the direct discrete solution for the hull-propeller-rudder system has become a reality. The sliding grid method is the most widely used method for solving propeller mesh motion. Typical grid distribution in the sliding grid method is shown in Fig. 2. The computational domain is divided into external domain and internal domain, in other words, static domain and dynamic domain. Propeller is contained in the dynamic domain, which is free to rotate relative to the static domain, and the interface between the two domains is used for flow field information transmission. The sliding grid method has the advantage of high precision and has been widely used in the simulation of hull-propeller-rudder coupling.

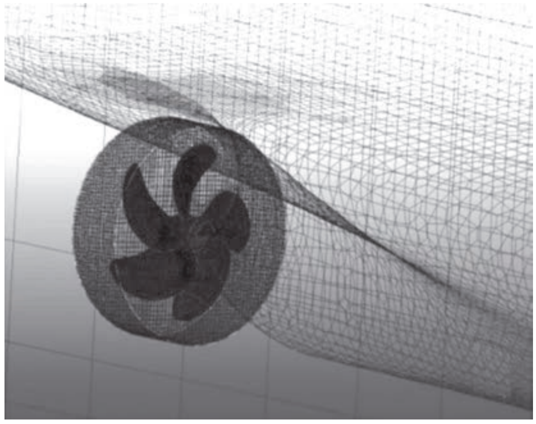

Fig. 2 Sliding grid distribution[14]

Lübke[12] used the sliding grid method to construct the hull-propeller coupling simulation model in commercial software CFX. Self-propulsion simulation was performed without free-surface calculation. Queutey et al.[13] simulated self-propulsion of KCS ship considering free-surface. Simulation was carried out using an in-house code ISIS-CFD. Calculation accuracy was improved by the adoption of adaptive mesh technique. Zhu et al.[14] simulated self-propulsion with the sliding grid method for the same ship model. They compared the wave heights of different positions near the model with the tests in detail. The above numerical studies using sliding grid method are all aimed at the self-propulsion of fixed ship model without rudder. Badoe et al.[15] performed the coupling simulation of hull, propeller and rudder based on sliding grid method. The interaction of rudder and propeller with different rudder angle was investigated in detail. Furthermore, Moctar et al.[16] constructed two sets of sliding grids for propeller and the rudder and carried out Zig-zag maneuver simulation for a ship model with two propellers and rudders.

Because of the advantage of being able to simulate unsteady problems with relatively high accuracy, the sliding grid method has been used to study more complex hull-propeller coupling problems considering propeller cavitation in recent years. Paik et al.[17] carried out numerical simulation of propeller cavitation after a full hull body to study the cavitation induced pressure fluctuation on ship hull. Schneer and Sauer model was applied to simulate the cavitation flow. Fujiyama[18] analysed the unsteady cavitation induced by propeller tip vortex after a ship hull. Results showed that the tip vortex cavitation causes the high-frequency component of pressure fluctuation.

Han et al.[19] proposed an integral calculation approach for numerical simulation of cavitating flow around a marine propeller behind the ship hull. And their numerical method was validated by comparison with experimental results. Long et al.[20] used the relative vorticity transport equation and particle trajectories method to study the cavitation flow around propeller behind a ship hull. Detailed flow structures were shown in their numerical results. Liu and Zhou[21] performed a fully hull-propeller-rudder coupling simulation with the consideration of cavitation. The pressure fluctuation on ship hull were found that has a high correlation with the second order partial derivative of cavitation volume.

Sliding grid method is indeed an effective method to study the hull-propeller-rudder coupling problems. The method has high numerical prediction accuracy and can accurately capture the interference of viscous flow field between ship, propeller and rudder. However, most of the researches at present using the sliding grid method are aimed at the ship model self-propulsion. It is still difficult to deal with large movements of ships such as manoeuvres.

1.3 *Overset grid method*

Compared with the body force method and the sliding grid method introduced in the previous sections, the overset grid method has the greatest advantage that it can realize large motion of multiple objects in the simulation of hull-propeller-rudder coupling problems. In the overset grid method, separate set of grids is built for each independently moving component. The grids can be structured or unstructured, as shown in Fig. 3. Flow field data exchange between different sets of grids is carried out by interpolation in the overlapping parts. By adopting the overset grid method, many scholars have carried out numerical simulation on the hull-propeller-rudder coupling problem during ship maneuvering.

Carrica et al.[22] performed URANS computations of standard maneuvers for a surface combatant at model and full scale, where steady turn and zigzag motion are simulated with overset grid method. The error between the calculated results and the experimental results is generally within 10%. The same solver is used by Mofidi and Carrica[23] to carry out the numerical simulation of the typical 10/10 zigzag maneuver and the modified 15/1 zigzag maneuver for KCS ship model. Detailed flow field under the interaction of hull, propeller and rudder was analysed. Above investigations are based on structured overset grid. Shen et al.[24] implanted overset grid module suitable for fully unstructured grids in OpenFOAM and developed a marine hydrodynamics solver naoefoam-SJTU. In this solver, the parallel calculating of the information interpolation in the overset grid is realized. Zigzag maneuver of KCS ship model was well simulated by this solver, which proves that satisfactory results can also be obtained by using reasonable distributed unstructured grid. Meng and Wan[25] used the solver successfully simulated the flow induced motion of an obliquely towed KVLCC2M model in deep and shallow water. Wang et al.[26] carried out self-propulsion and steady turning simulation for fully appended ONR ship model. The predicted self-propulsion point and turning circle are in good agreement with experimental data, which further verifies the accuracy of the solver. Self-propulsion in shallow water was then simulated by Wang et al.[27] Results showed that the shallow water condition has significant effect on the ship advancing with rotating propeller.

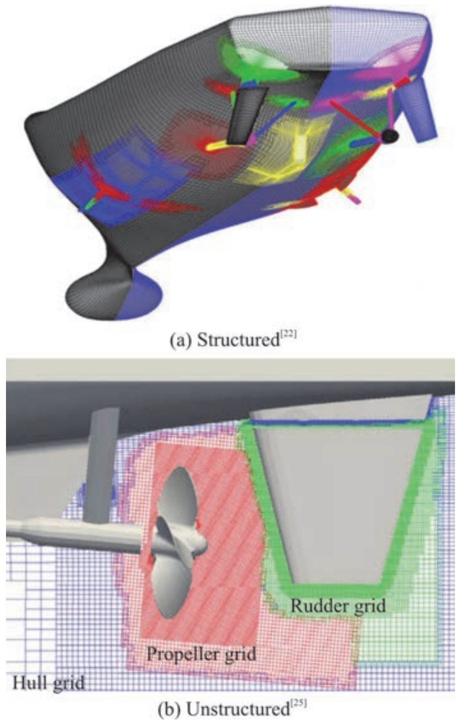

Fig. 3 (Color online) Overset grid arrangement

Recently, overset grid method was used to investigate more complex problems such as ship maneuver in waves and ship course keeping. Wang and Wan[28] adopted a self-developed 3-D wave tank module to simulate ONR tumblehome ship model under self-propulsion in head waves. The predicted ship motions were compared with experimental results. Furthermore, they extended the simulation to many different wave conditions[29] and course keeping was also simulated at the same time[30]. These simulations have proved that the overset grid method has a strong advantage in simulating the coupling of hull, propeller and rudder under various complex motions.

2. Wave-floating structure coupling

Waves are the main source of external loads on ships and ocean engineering structures. Seakeeping involves a series of problems, such as the motion responses of ship and platforms, ship added resistance, slamming, green water, etc., which have an important impact on the safety and effectiveness of ship and ocean engineering structures. Many numerical methods have been proposed to predict the wave-structure coupling problems, which can be classified as potential flow method and viscous flow method. The potential flow method is obviously faster. Although the viscous effect is neglected, relatively accurate results can be obtained with reasonable damping coefficient[31]. boundary element method (BEM) is the most popular potential flow method. The advantage of the viscous flow method is that it can deal with more complex nonlinear and viscous effects. And the viscous flow method can be further subdivided into mesh method and meshless method. Numerical investigations based on the above methods will be introduced in the following sections.

2.1 *Boundary element method*

In the boundary element method, the solution only need–to be carried out on the boundary, which reduces the dimension of solving problem. Thus it has been widely applied. The simplest is the constant boundary element method (CBEM)[32]. A smooth structure surface is replaced by many quadrilateral or triangular plane elements, and the unknown velocity potential and its normal derivative are assumed to be constant in each element. This method is simple in dealing with complex curved model. Therefore, the high-order boundary element method (HOBEM) was proposed and adopted[33]. Coordinates, velocity potential and other physical quantities in elements are expressed as functions of node values by quadratic shape functions. The potential function is continuous over the entire surface, which can obviously improve the efficiency and accuracy. Because the HOBEM is piecewise interpolation, the spatial derivative of the velocity potential is continuous only within the elements, but cannot guarantee the continuity between the elements cannot be guaranteed, which limits the further calculation of higher order problems. On the basis, the idea of spline function is added into the process of solving the integral equation by the boundary element method[34], forming the B-spline HOBEM method. In order to meet the computational requirements of the interaction between super-large structures and waves in recent years, fast multipole method[35] and self-adaptive projection method[36] are introduced into the BEM method to further improve the computational efficiency and accuracy.

The calculation of wave-structure coupling by BEM method can be divided into frequency-domain theory and time-domain theory. In the frequency-domain theory, physical quantities of the wave field are assumed to change periodically with time. Second or third order solutions can be obtained by means of perturbation expansion in nonlinear frequency theory. For example, Chau and Taulor[37] carried out a detailed analysis of the second-order diffraction problem of a uniform vertical circular cylinder in regular waves. Malenica and Molin[38] calculated third-harmonic wave diffraction by a vertical cylinder in finite depth water. However, it is difficult to develop this method to higher order. In addition, limited by the basic assumptions, the frequency domain method is usually only applicable to periodic steady state problems. For nonlinear problems, scholars prefer to use the time-domain theory.

Despite the increased computation, the time-domain method has the advantage that can handle the coupling of waves with structures with arbitrary motion and is effective in solving nonlinear problems. Isaacson and Cheung[39] were the first to propose second order time-domain theory, in which first and second order boundary value problems are established based on Taylor series expansion and perturbation expansion. Many further improvements were proposed during recent years. Duan et al.[40] developed second-order Taylor expansion BEM method, which can provide accurate solution of the second-order velocity potential and sum-frequency wave loads for the floating bodies with sharp corners. Xu et al.[41] proposed a hybrid radiation condition, which is the composition of the multi-transmitting formula method and damping zone method, to minimize the wave reflection. In the condition of large wave amplitude or large movement of the structure, the second or higher order radiation and diffraction analysis based on perturbation expansion will be a failure. Therefore, it is necessary to carry out fully nonlinear calculation. The earliest and most famous fully nonlinear theory was the mixed Eulerian-Lagranian (MEL) method[42]. Grilli et al.[43-44] combined 3-D HOBEM and MEL method and performed numerical calculation of the interaction between nonlinear waves and structures. The wave overturning and breaking were well predicted. Feng[45] developed an algorithm which combined Rankine source method with Mixed Euler-Lagrange method. Both 2- and 3-D forced body oscillatory motion problems were studied and the predicted results were proved to be reliable. Zhou et al.[46] developed a 3-D fully nonlinear numerical wave tank (NWT) based on the MEL method. By typical applications wave evolution and harmonic decomposition during propagation over a submerged bar, the NWT was proved to be more accurate than the Boussinesq model. The scale of ocean structures are becoming larger and larger in recent years. In order to overcome the large computational cost caused by full matrix, Teng and Gou[47] discussed fast algorithms with high speed and low storage, including fast multipole method (FMM) and precorrected fast Fourier transform (pFFT). Principles and applications were introduced in detail in their paper.

2.2 *Viscous flow mesh method*

Numerical method based on potential flow has the most obvious disadvantage that viscous effect cannot be solved. In recent years, turbulent and viscous effect under complex wave conditions and their influence on structures are getting more and more attention. Therefore, numerical simulation methods based on Navier-Stokes equations and discretized by finite volume method and finite difference method are gradually applied to the investigation of wave-structure coupling problem. In order to accurately simulate the wave evolution and break caused by slamming, volume of fluid (VOF) and Level Set are two mainly adopted interface capturing method.

In the VOF method, a volume fraction α is used to identify different phase. There is an interface if α is between 0 and 1. VOF method has been adopted in various complex wave-structure coupling problems because of the good mass conservation and computational efficiency. Castiglione et al.[48] carried out numerical simulations to predict the seakeeping characteristics of a high-speed multi-hull vessel in high sea states. By the comparison of wave pattern between potential flow strip theory and viscous flow simulation, it was proved that the viscous simulation is more accurate and allows the detection of nonlinear effects. Shen et al.[49] used the in-house marine hydrodynamics solver naoe-FOAM-SJTU to investigate motion response and added resistance of Wigley ship in head waves. In the condition of large wave amplitude, the strong nonlinearity of wave added resistance can be well predicted. Ye et al.[50] adopted the same solver to study the ship bow slamming phenomenon for the S-175 ship model under regular head waves. Free surface at four instants over an encounter period can be seen in Fig. 4, in which wave breaking around the ship bow can be seen. Shen et al.[51] further investigate the interaction between long-crest irregular waves with three different ship models. Under this complex strong nonlinear wave conditions, the advantage of CFD are further heighted in contrast with potential flow calculation. Liu et al.[52] adopted a relaxation technique for the wave generation and absorption to avoid wave reflection from the boundaries and investigated wave forces and motions of DTC ship in oblique waves. In addition to the seakeeping of ships, the interaction between waves and ocean structures such as offshore platforms is also the focus of engineering attention. Danmeier and Seah[53] used a VOF based two phase viscous flow solver ComFLOW to study the wave run up for a gravity based structure. Wave breaking during the run up can be reproduced basically in the simulation. Wave run up phenomenon was also studied by Cao and Wan[54] The relationships of the wave run-up height with the incident wave height and cylinder radius are analysed. Liu and Wan[55] studied the motion responses of a complex ocean observation platform in different wave directions and length. Results showed that the heave motion is sensible to the change of wave length. Furthermore, Liu et al.[56] added mooring system in the solver and performed numerical simulation of a submersible platform in waves with 12 mooring lines. The coupled effect of wave, platform and mooring system was analysed in detail. Zhuang and Wan[57] took advantage of a benchmark case, focused waves acting on a fixed FPSO-shaped body to validate their numerical method and studied the effect of wave steepness.

Compared with VOF method, level set method has the advantages of simple calculation and easy implementation. A distance function is defined in level set method, whose zero level is the location of the air/water interface. By assuming that the density and viscosity of the air are negligible and the air/water interface remains at all times at atmospheric pressure, Carrica and Wilson[58-59] developed a single phase level-set algorithm in their solver CFDShip-Iowa. The solver was first validated in detail by comparison with experiment data. Then, the coupled pitch and heave were studied for a DTMB 5512 ship model in regular head waves. A significant reduction on the transfer function was observed for both pitch and heave in large amplitude waves, which indicate strong nonlinear effect. Besides, added resistance were also reported. Different from the single phase level set used by Carrica and Wilson[58-59], Bish et al.[60] adopted a two phase Level Set method and developed a CFD solver REEF3D. The performance of the solver were tested by several benchmark applications. Results showed that complex two phase phenomena such as wave breaking can be well simulated by their solver. At the same time, Bish et al.[61] adopted the solver REEF3D to investigate the breaking wave interaction with tandem cylinders under different impact scenarios. Typical figures of the wave evolution can be seen in Fig. 5. Results indicated that after the wave pass the first cylinder, water jet induced by the separation of the wavefront and overturning wave crest have a significant effect on the second cylinder. Chella et al.[62] used the same solver to study the breaking solitary waves and breaking wave forces on a vertically mounted slender cylinder over an imper-

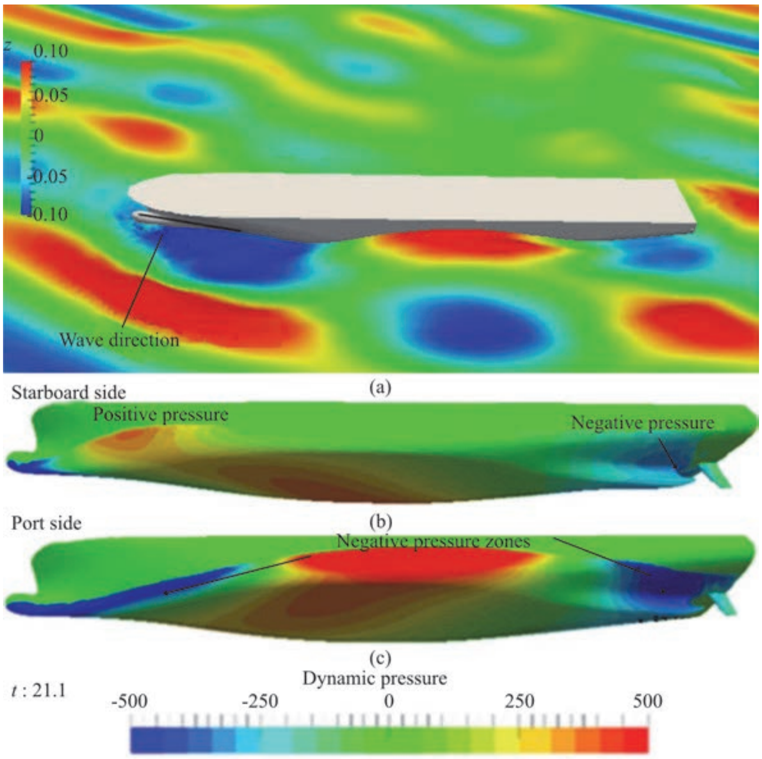

Fig. 4 (Color online) VOF simulation of DTC ship in oblique waves[52]

meable sloping seabed. They found that for a given slope, a solitary wave propagates further up on the slope and breaks in a shallower water depth than periodic waves. Teng et al.[63] developed a viscous fluid numerical wave tank and carried out simulation on the interaction between waves and a submerged horizontal circular cylinder undergoing forced oscillation. By comparison with potential results and experimental results, the viscous effects were found obviously under large motion. Viscous vortex was regarded as the underlying mechanism.

2.3 *Viscous flow meshless method*

There is no doubt that wave breaking and droplet splashing are the most difficult points in the problem of wave-structure coupling. In the viscous flow methods, the meshless method has inherent advantages over the mesh method in the simulation of strongly nonlinear free surfaces. In the meshless method, the fluid particles are tracked by the Lagrangian framework. When the free surface is broken, the fluid particles splash in the form of Lagrangian particles, which does not require interface reconstruction. Therefore, high precision can be achieved. The most commonly used meshless method includes smoothed particle hydrodynamics (SPH) method and moving particle semi-implicit (MPS) method.

The "smooth" in the SPH method refers to that the physical variables at each point is determined as a weighted average of values in a local region. The SPH method has undergone a lot of optimization since it was proposed. Khayyer et al.[64] proposed a corrected incompressible SPH (CISPH) method in order to accurately track the water free surface in wave breaking problems. Corrective terms were introduced to guarantee the exact calculation of linear velocity fields and ensures the preservation of angular momentum. Breaking of solitary waves on a plane slope were simulated to validate their algorithm. Rudman and Cleary[65-67] carried out a serious of studies about the interaction between rough waves and a tension leg platform using SPH method. Mooring system was also considered in their simulation. Initially, they studied the effect of wave height with constant wave direction on a semisubmersible platform with two mooring systems[65]. Results showed that the heave and surge responses are more sensitive. Next, they considered the effect of impact angle for rogue wave[66] and found that the impact force is largest when angle is 45°. Recently, they focused on the mooring line design under the effect of rough waves[67]. Typical results of their SPH analysis can be seen in Fig. 6. Ren et al.[68] proposed an improved weakly compressible smoothed particle hydrodynamic (WCSPH) method and performed numerical simulation of the wave interactions with a submerged rubble-mound breakwater. Their algorithm was proved that can well simulate the evolution of turbulent flow.

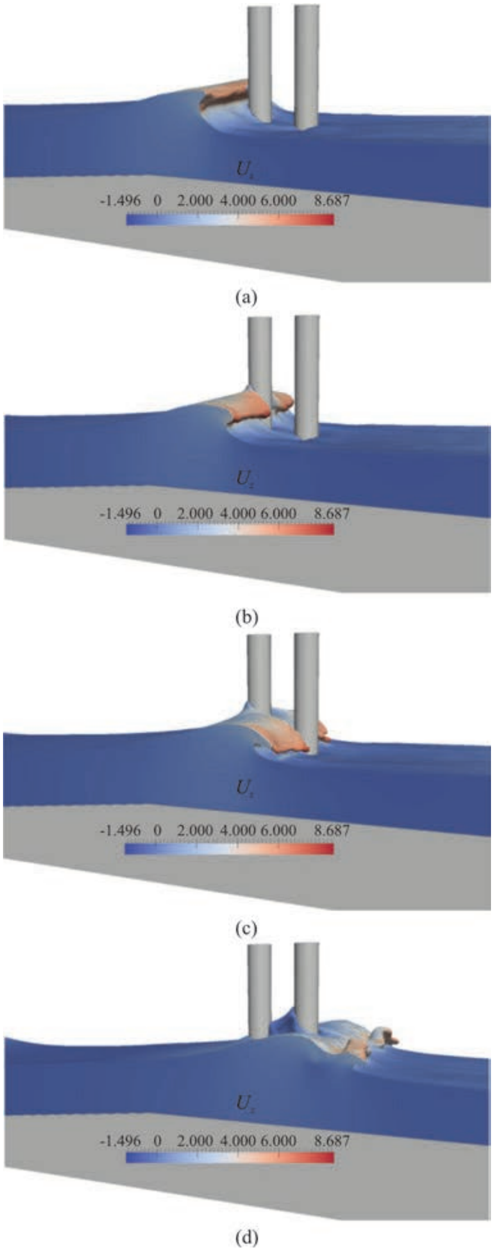

Fig. 5 (Color online) Free-surface capturing by level set method for the wave-tandem cylinders interaction problem[61]

MPS method was originally proposed for incompressible flow. Fluid pressure field is solved by a semi-implicit algorithm. Although the solution of the pressure poisson equation is time-consuming, a relatively large time step can be accepted in this method. Therefore, MPS method is becoming more and more popular in the simulation of wave-structure coupling problems. Shibata et al.[69] simulated nonlinear waves by the original MPS method. A transparent boundary condition was developed to generate progressive waves. By comparing with the experiment, the method is proved to be effective in wave generation. However, there was no structure in the numerical wave tank. Zhang et al.[70] introduced high order Laplacian model into the original MPS method to improve the unphysical pressure oscillation and developed an in-house solver MLParticle-SJTU. Interaction between regular waves and free roll motion of a 2-D floating body was investigated. Wave forces and motion responses of the body were in good agreement with experimental results[71]. Rao and Wan[72] combined the improved MPS method with Finite Element Method and investigated fluid-structure interaction (FSI) problems with waves. Solitary wave impact on a flexible plate was simulated and the evolution of the free surface can be seen in Fig. 7. The same method was adopted by Zhang et al.[73] to study the effect of regular waves. By comparison between rigid case and elastic case, the impact duration time when slamming n a flexible structure was found to be less than half of that regarding the rigid case.

3. Aerodynamic-hydrodynamic coupling

Floating offshore wind turbine has the tendency to become the main part of the future offshore wind power generation technology. The movement of each part of the floating offshore wind turbine is independent and affects each other. On the one hand, the 6DOF movement of the platform under the combined action of wind waves and currents will produce wake distortion, wake deformation, diffusion, causing drastic changes in the flow field around the turbine blade, which may seriously lead to fatigue damage. On the other hand, the aerodynamic characteristics of the blade will intensify the movement response of the supporting platform. The working principle of the floating fan system is very complicated, which is the result of the coupling of the aerodynamic performance of the blade and the hydrodynamic performance of the supporting floating body. At present, the main method for numerical study is the combination of blade element momentum (BEM) method with potential flow theory. With the development of computer technology, fully coupled CFD method is gradually applied to the design and evaluation of floating offshore wind turbine. And on the basis, actuator line model is adopted to model the blades to improve the computational efficiency.

3.1 *Coupled analysis of BEM and potential flow theory*

BEM theory, as one of the most classic methods to study the aerodynamic characteristics of wind turbines, was first proposed by Glauert[74]. The basic idea of this method is to transform the flow around a

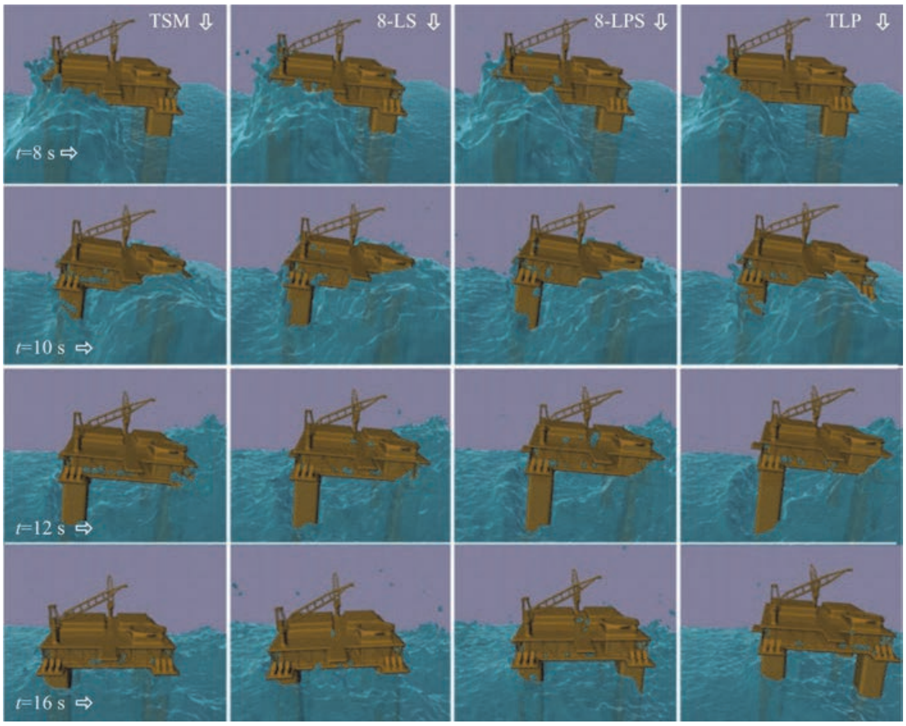

Fig. 6 (Color online) SPH simulation results for a tension leg platform (TLP) platform in rough waves[64]

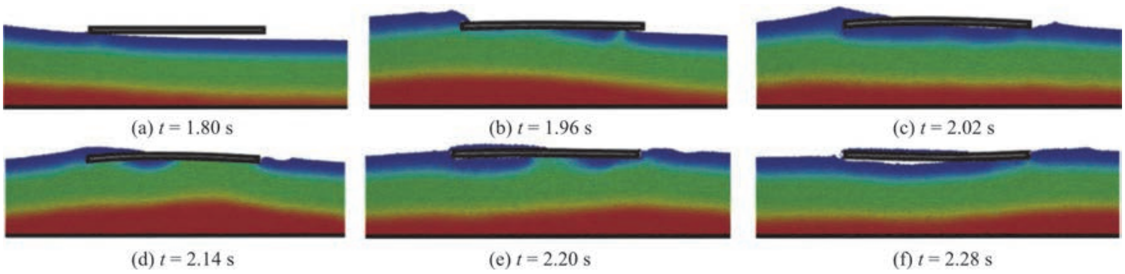

Fig. 7 (Color online) MPS simulation results for an elastic plate in waves[72]

blade in a complex 3-D blade into a 2-D flow around an airfoil. Relevant data of blade profile, such as lift coefficient, drag coefficient and incident angle of attack at different sections, are used to analyse the distribution of different aerodynamic loads along the blade. This simplified method requires correction of the results to obtain more accurate aerodynamic performance characteristics, including blade tip loss correction, dynamic stall correction, hub loss correction and yaw wake correction. On the other hand, potential flow theory assumes that waves acting on floating platform can be regarded as uniform, incompressible and non-viscous ideal fluid, which can fast predict the hydrodynamic performance of the supporting platform.

Kim and Sclavounos[75] simulated fully coupled large amplitude simulations of theme offshore structures in water depths of up to 10^4 feet. The linear and slow-drift responses and the mooring-line/tendon dynamic tension of a Spar and a TLP in wind, waves and currents were presented. Finally, they developed the program into a series of SML software[76-78] that can predict and analyse the aerodynamic, hydrodynamic and mooring fully coupled dynamic responses of floating offshore wind turbine systems. Nielsen et al.[79] combined the advantages of SIMO/RIFLEX and HAWC2 software to simulate the fully coupled motion response of the floating wind turbine system under the action of different wave frequencies and wind speeds. HAWC2 was used for dynamic analysis of the load on the turbine, and SIMO/RIFLEX was used to solve the structural elastic deformation and the hydrodynamic performance of the platform. By integrating the main functions of FAST, AeroDyn, HydroDyn software, Jonkman[80] formed a tool to analyse the aerodynamic and hydrodynamic coupling dynamic characteristics of the floating wind turbine system. And based on this tool, they analysed the aerodynamic and hydrodynamic coupling dynamic response characteristics of the barge floating wind system. Bae et al.[81] studied a mini TLP-type floating offshore wind turbine that can be used in deeper areas. Both CHARM3D and FAST software were used in their analysis. The results showed that the probability of damage to the turbine blade structure is increased under high frequency shock loads. Bachynski et al.[82] combined SIMO solver, AeroDyn solver and RIFLEX solver to study the influence of the angle between wind and wave on the overall performance and structural fatigue characteristics of the floating wind turbine. Ma et al.[83] adopted FAST solver to analyse the dynamic responses and mooring loads for a spar-type floating wind turbine induced by coupled wind and wave loads. A fully coupled time-domain aero-hydro-servo-elastic simulation of a tension-leg-platform (TLP) floating wind turbine is conducted by Shen et al.[84]. Time series results are obtained and analysed to study the coupled dynamic responses of the TLP floating wind turbine. Zhou et al.[85] also used the FAST software to carry out aero-hydro-servo-elastic simulation for a new 6 MW spar-type floating offshore wind turbine. The second-order wave force were calculated by SESAM. Qu et al.[86] established an analytical model coupling wind turbine, tower, floating foundation and mooring line for a novel floating offshore wind turbine with a weathervane function and moored by the single-point mooring system. Most of the current wind turbines are the type with horizontal axis. Guo et al.[87] performed a numerical investigation on the surge-heave-pitch coupling motions of the Φ type vertical axis wind turbine (VAWT) supported by the truss Spar floating foundation. The 2P (twice-per-revolution) responses were found more significant in the irregular wave and steady wind conditions.

3.2 *Fully coupled CFD*

The main advantage of coupled analysis of BEM and potential flow theory is high computational efficiency. However, due to the neglect of the viscous effect of the surrounding flow field and the influence of flow separation, the performance of the flow field of the platform and the blades cannot be captured. In consideration of the complexity of aerodynamic and hydrodynamic coupling of floating offshore wind turbine system, CFD numerical simulation tools with real scale and real model have become a hot topic in research.

Ren et al.[88] adopted the commercial software FLUENT with the User Defined Function (UDF) to carry out CFD analysis of a 5 MW floating wind turbine system supported by a TLP under coupled wave-wind loads. The numerical results were in good agreement with experiment. They concluded that both the hydrodynamic forces and aerodynamic forces have obvious effect on the motion responses of the system. The biggest simplification of their research work is that only the surge motion was considered. Liu et al.[89-90] carried out further works, they considered three degrees of freedom responses of the floating structure, which are surge, heave and pitch. A built-in sliding grid technique in OpenFOAM termed arbitrary mesh interface (AMI) was adopted to cope with the relative motion. The coupling effect between the modules of OC4 semi-submersible floating wind turbine had been successfully analysed. However, it is necessary to set up multiple sliding zones to meet the requirements of multiple degrees of freedom movement, which also greatly restricts the application and development of sliding grid technology in the CFD simulation of floating wind turbine system.

In order to overcome the challenges in mesh motion, overset technique was adopted for the CFD simulation of aerodynamic-hydrodynamic coupling problems. Quallen and Xing[91] performed a full-system, two phase simulation with the consideration of wave and wind loads for an OC3 spar-type FOWT. The CFD results were compared to the results using FAST software. The trends of the motions agreed well with those of the NREL results, but showing 25% less about mean surge, which was believed due to the constant drag coefficient adopted in FAST. Then they combined a variable-speed generator-torque controller with the two phase CFD solver CFDShip-Iowa V4.5 and simulated the 5 MW floating offshore wind turbine[92]. Figure 8 shows the 3-D views of turbine and vortical structures in their simulation. Tran and Kim[93-94] used the commercial computational fluid dynamics software StarCCM+ to simulate the aerodynamic and hydrodynamic coupling of OC4 semi-submersible floating wind turbine, and applied the overset grid technology to deal with the six degree of freedom motion problems. Their numerical simulation results were in good agreement with those of FAST. Cheng and Wan[95] employed their in-house CFD code naoeFoam-SJTU to perform fully coupled aero-hydrodynamic simulation of the OC4 Phase II semi-submersible floating offshore wind turbine.

3.3 *ALM-CFD*

The inevitable problem of fully coupled CFD simulation is the high calculation cost and long calculation time. The calculation of real blade rotation is the most time-consuming part of the simulation. Therefore, the actuator line model (ALM) method was proposed, in which the blades and tower are not modelled. Each blade and tower are both divided into a series of discrete actuator points. Then the body forces

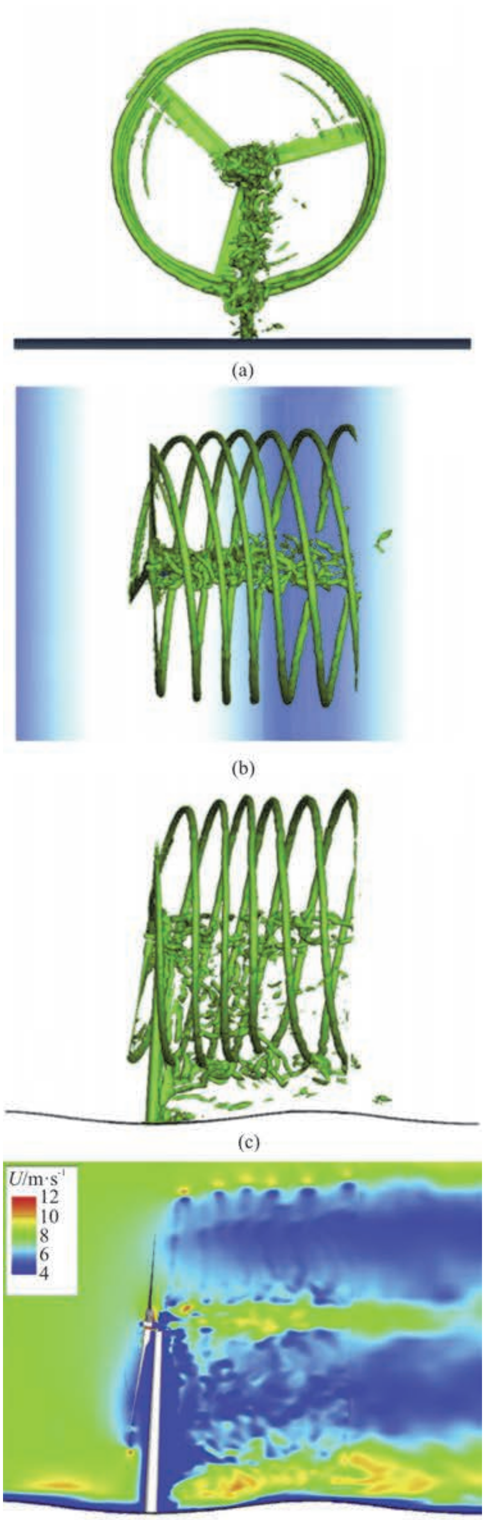

Fig. 8 (Color online) 3-D views of turbine and vortical structures in fully coupled CFD simulaton[92]

acting on the points can be obtained according to 2-D airfoil profile data. The forces on each point are smoothed to the effectible cells using the special smooth technique. The result of these body forces on the cells are treated as a source term of the NS equations. Typical ALM-CFD results can be seen in Fig. 9, the turbine wake can be calculated well although there are no blades. Because the information of the flow field can be obtained without solving the complicated flow on the blade surface, the actuator line model has been widely applied, especially in the simulation of wind farms[96].

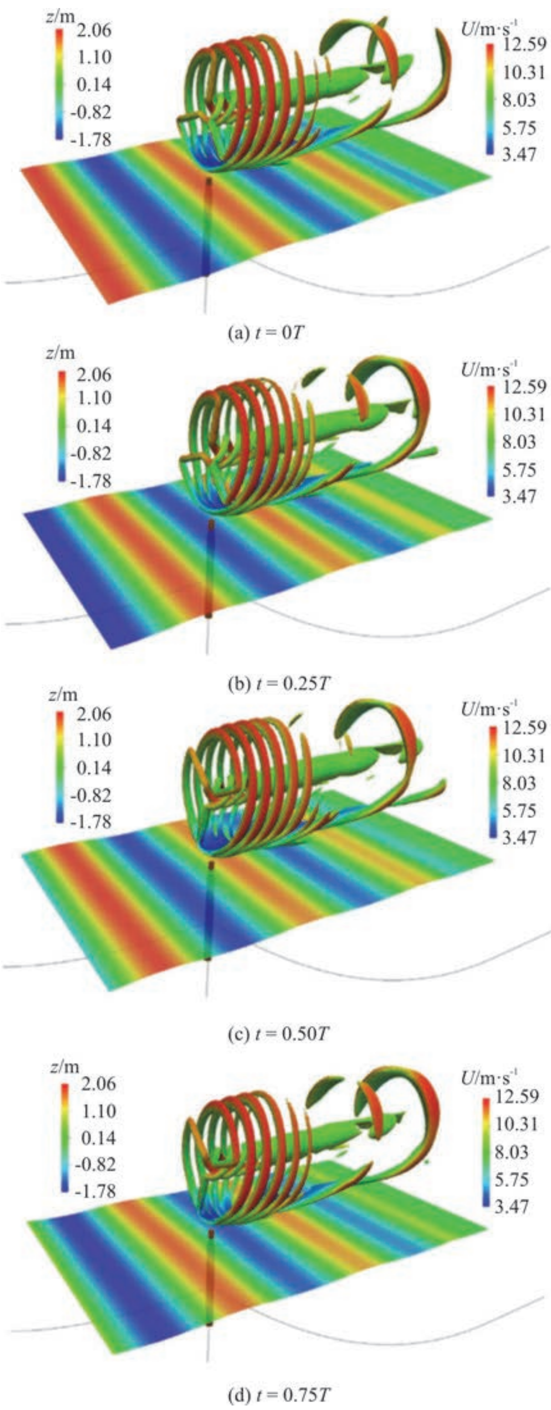

Fig. 9 (Color online) Vortex structure of the rotor calculated by ALM-CFD method[96]

Cheng and Wan[97] made detailed comparison of the numerical simulation results obtained by using fully coupled CFD method and ALM-CFD method. Simulated model is a semi-submersible floating offshore wind turbine system. First for the computational efficiency, time consumption in the simulation using ALM-CFD method is just 1/3 of that using fully coupled CFD method. At the same time, the difference of aerodynamic thrust and platform motion responses between two methods is about 5%-10%. These results further proved the advantage of ALM-CFD in engineering application. Huang et al.[98] carried out coupled aero-hydrodynamic analysis on a floating offshore wind turbine under extreme sea conditions. The maximum wave height in the simulation reached 10m. They concluded that the mean value of the aerodynamic loads is mainly dominated by the wind loads, whereas the fluctuation range of the aerodynamic loads mainly depends on the wave loads. Huang et al.[99] then performed simulations with different complexities. First, the wind turbine was parked. Second, the impact of the wind turbine was simplified into equivalent forces and moments. Third, fully coupled dynamic analysis with wind and wave excitation was conducted. Cheng et al.[100] made a detailed verification analysis of their solver FOWT-UALM-SJTU and proved that the solver has satisfactory efficiency and accuracy. Ning and Wan[101] adopted Large Eddy Simulation to generate an atmospheric boundary layer flow. Blades of turbine were modelled by ALM and the effect of turbulent inflow on the wake was analysed in detail. Huang and Wan[102] used the FOWT-UALM-SJTU solver to investigate the coupled aero-hydrodynamic performance of a spar-type FOWT composed of the NREL 5-MW wind turbine and the OC3-Hywind spar platform.

4. Fluid-structure coupling

Under the complex environment in deep sea, fluid-structure coupling problems caused by violent flow are very common, such as wave slamming on ships and sloshing in a partially-filled liquid tank of moving ships. These fluid-structure coupling processes are often accompanied by violent free surface changes, and the corresponding wave load can induce the bending and torsion deformation of the structures, as well as the vibration, instability and failure of the local structure. Under this background, many coupled methods have been developed to simulate the fluid-structure coupling problems. In the following section, three mature methods focusing on ship and ocean engineering field will be introduced.

4.1 *FVM-FEM*

In the last decade or so, researchers have used different methods for numerical analysis of fluid-solid coupling problems. Currently, the most common method is still integrating finite volume method (FVM) and the finite element method (FEM), which combines computational fluid dynamics (CFD) and computational structure dynamics (CSD). In the study of Oberhagemann et al.[103], the structural response of a liquefied natural gas carrier (LNG) under stern slamming was predicted numerically as Fig. 10 shows, the structural response of a liquefied natural gas carrier (LNG) under stern slamming was predicted numerically, through the viscous fluid solver in the commercial finite element software ANSYS. But the longitudinal stresses and shear stresses caused by quasisteady wave bending were uncritical. This study ignores the influence of structural deformation on the viscous flow field, therefore, it belongs to the category of one-way fluid-solid coupling. Maki[104] used open source CFD library OpenFOAM to simulate hydroelastic impact of a wedge-shaped body in two dimension onto a calm free-surface. The method predicts the hydroelastic response in the time domain. However, it requires approximation of the flexural added mass and it is still one-way coupling. Pari and Maki[105] studied the entry and exit of flexible bodies through an air-water interface using a tightly coupled fluid-structure interaction solver. The numerical simulation results show that the wedge-shaped objects with different section thicknesses exhibit high-frequency vibration during the movement. It is shown that hydroelastic effects cannot be ignored for entry and exit problems. That means the two-way coupling is needed. Lakshmynarayanana et al.[106] used a two-way coupling between Reynolds average numerical simulation (RANS) and finite element software (FES) to predict response of flexible floating bodies in regular waves, through commercial software such as StarCCM+ and ABAQUS. The structure of the flexible barge is modelled as beam. For S-175 container ship model, the computational results of hydroelastic predictions are compared with experimental measurements. Qin[107] considered the fluid-structure interaction (FSI), by combining the fluid domain and the structure domain discretized by the finite element method (FEM) in a fully-coupled way, studying the dynamic response of a horizontal plate dropping onto nonlinear freak waves. A self-developed solver is used to simulate the freak waves numerically, in the numerical wave tank built in 2-D manner.

4.2 *Meshless method*

The fluid-structure coupling problems in ship and ocean engineering field are usually caused by the interaction between waves and structures. Because of the obvious advantages of meshless method in simula-

Fig. 10 (Color online) Hull girder modes at eigenfrequency of 3.59Hz for wave[103]

ting the drastic change of free surface, scholars gradually introduce this method into the numerical simulation of ship and ocean engineering fluid-structure coupling problems. The most commonly used methods include the combination of smoothed particle hydrodynamic with finite element method (SPH-FEM) and the combination of moving particle semi-implicit with finite element method (MPS-FEM), At the same time, there are also some studies that used meshless method for both fluid field solving and structure field solving.

Vuyst et al.[108] were the first to apply the SPH-FEM combination method to the fluid-structure coupling problem with free surface. However, their results did not give a clear structural deformation response. Antoci et al.[109] carried out both experimental study and numerical simulation for fluid-structure coupling between channel discharged water and an elastic gate. They successfully simulated the phenomenon in experiment by using SPH-FEM method, which fully showed the application prospect of this method in the simulation of fluid-structure coupling problems. Yang et al.[110] proposed a loosely coupled SPH–FEM model, which was validated by two benchmark FSI problems. Their typical results can be seen in Fig. 11. Hu et al.[111] found that neighbour searching and contact searching in the coupled SPH-FEM model are extremely time-consuming. Therefore, they proposed a novel Striped-PIB (S-PIB) searching algorithm to improve the computational efficiency. Through numerical simulation of the interaction between dam break wave and elastic structure, the improved algorithm can enhance the calculation efficiency by over 56%. Long et al. improved many key problems in the SPM-FEM method. First, the particle-element contact algorithm was incorporated into the SPM-FEM model to adjust positions and normal velocities of slave particles and master nodes by conservation of linear momentum and angular momentum[112]. In addition, they proposed a new ghost particle method to treat the arbitrary boundary of FSI problem[113]. Several standard numerical simulations were performed to prove that their improvements are effective.

In contrast with SPH method, MPS is a completely incompressible flow field calculation method. Therefore, MPS method is more suitable for the calculation of ship and ocean engineering problems.

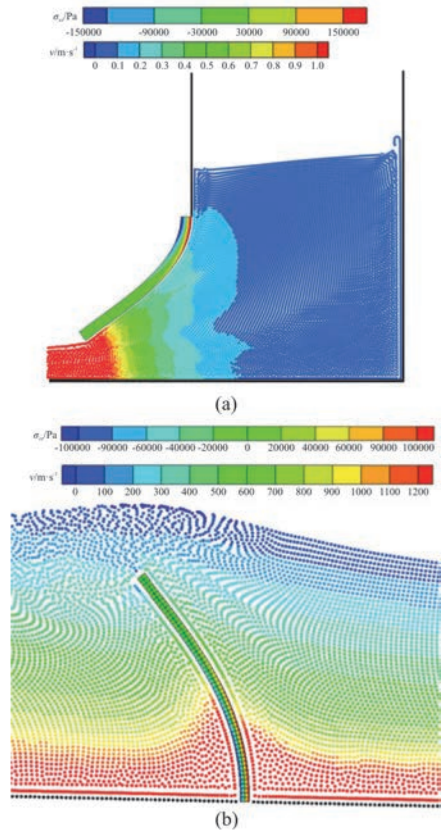

Fig. 11 (Color online) Fluid-structure interaction simulation by a loosely coupled SPH-FEM model[110]

Lee et al.[114] was the first to combine MPS with FEM and investigated liquid sloshing in an elastic tank. MPS and FEM solving module are executed serially during each time step. How to express the surface of deformed structures accurately is a key problem in the study of fluid-structure coupling with meshless method. In general, the surface of structures are presented by multilayer particles. However, when the structure deforms, the inhomogeneity of local particles will lead to the decrease of the accuracy of surface expression. Mitsume et al.[115] adopted a new structure surface expression method in their MPS-FEM simulation. The interface between fluid and structure domains was handled by a MPS polygon wall rather than wall particles. A dam break problem with an elastic obstacle was simulated to validate their improvement. It is worth noting that pressure field in their results showed obvious rough characteristic. Hwang et al.[116] adopted an improved semi-implicit MPS method to simulate the flow field. From the simulation results of the interaction process between the shaking wave of the tank and the suspended elastic beam, it can be seen that although the physical phenomenon has obvious nonlinear characteristics, the pressure in the flow field still maintains a smooth distribution state. Zhang et al.[117] developed an in-house particle solver MLParticle-SJTU based on improved MPS method. FSI problems of sloshing with elastic baffles were numerically studied. Deformations of the baffles, including the linear and nonlinear responses, were quite coincident with the experimental results. Then, Zhang and Wan numerically analysed elastic structural response due to the impact loads of sloshing flows[118] and dam-break flow[119] by the same solver. Typical sloshing flow field and deformations of the elastic wall can be seen in Fig. 12. Chen et al.[120] studied the influences of structural elasticity on free surface in a liquid sloshing problems. The evolution of free surface can be well reflected by particle motion. Sun et al.[121] applied modified MPS and modal superposition methods to compute the hydroelasticity response of a cylindrical shell and a ship hull involving violent free surface deformation and slamming.

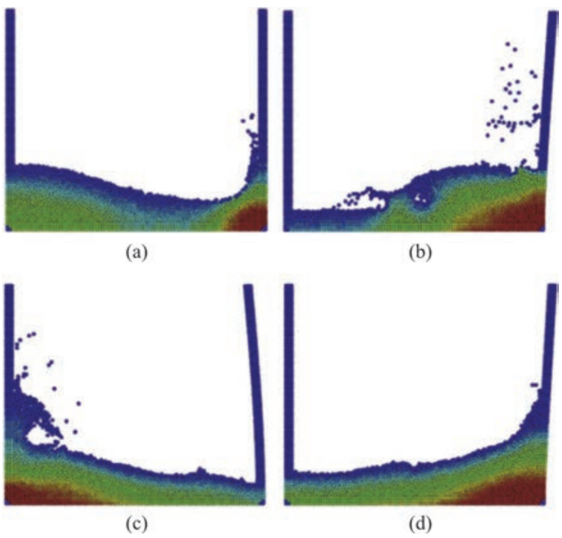

Fig. 12 (Color online) Flow field and deformations of the elastic wall solved by MLParticle-SJTU[118]

Structure field solving in all of the studies introduced above are based on FEM method. There are still some studies in which both fluid field solving and structure field solving are based on meshless method. Han and Hu[122] proposed a new numerical modelling for FSI problems based on SPH method. They improved the disadvantage of inconsistency in SPH for the structure filed solving by introducing a correction matrix. Khayyer et al.[123] developed an enhanced fully-Lagrangian meshless computational method for FSI problems based on MPS method. The numerical methods were shown to possess appropriate and comparable levels of stability and accuracy. A more detailed review of the meshless method in fluid-structure coupling problems can be found in the introduction of Zhang et al.[124]

4.3 *Strip method*

Some special problems in ship and ocean engineering field have the characteristic that there is a large scale in one direction. The most typical is the marine riser. In order to avoid the problem of large resource consumption and long calculation time for full 3-D numerical simulation of this kind of problem, strip method is proposed. As Fig. 13 shows, several sections along the span-wise direction of the riser are selected, and CFD method is applied to solve the hydrodynamic forces on each section. Because of its high computational efficiency, strip method has become a mainstream numerical method for predicting the vortex induced vibration (VIV) response of slender flexible riser.

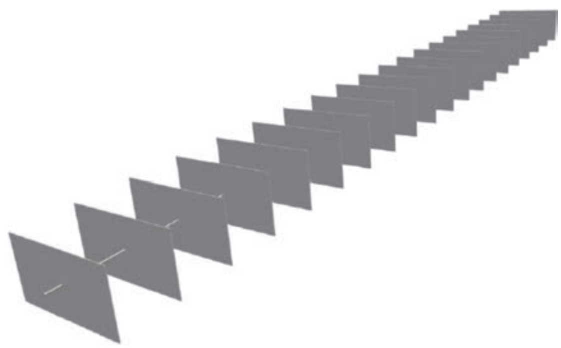

Fig. 13 Strips located equidistantly along the span of the flexible riser[130]

Herfjord et al.[125] developed a fluid-structure coupling calculation code based on strip method. The flow field was calculated using the CFD code NAVISM, while structural dynamics of riser is calculated by USFOS code. In each time step, a coupling module is used to carry out the data interaction between the two modules, so as to realize the vortex-induced vibration simulation of the deep-sea riser. Yamamoto et al.[126] also carried out a study on vortex-induced vibration of deep-sea riser based on strip method. In the calculation of flow field section, a discrete vortex method (DVM) suitable for simulating 2-D incompressible viscous fluid was adopted. Duanmu et al.[127-128] carried out numerical studies on multi-modal vibrations of vertical riser in uniform currents, shear currents and step currents. The vibration modes in both in-line (IL) and cross-flow

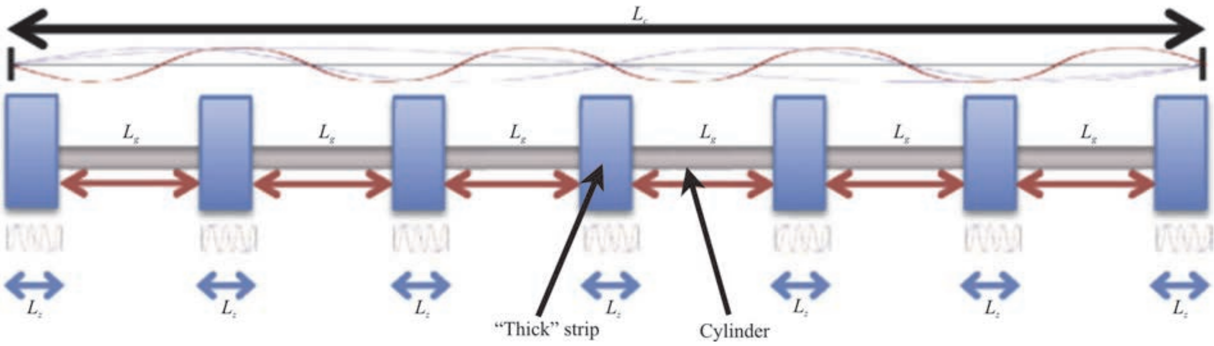

Fig. 14 (Color online) Schematic diagram of the thick strip method[131]

(CF) directions were accurately predicted. Fu et al.[129-130] investigated the vortex-induced vibrations of a flexible cylinder in an oscillatory flow. Computational results were compared with experimental data and further analysed. According to the basic assumptions of strip method, flow field on each section is 2-D. However, there is an inherent disadvantage that the 3-D effect of turbulent vortex cannot be simulated. Therefore, Bao et al.[131] proposed thick strip model for the simulation of VIV problems. The fluid domain was divided into many strips with certain thickness as Fig. 14 shows. Numerical simulation shows that the thick strip method can improve the accuracy of the results. More recently, Bao et al.[132] investigated vortex-induced vibration of a long flexible tensioned riser subject to uniform currents by the thick strip model. Variation of vortex shedding is visualized in the local wakes, indicating the coupling of "thick" strips through the structural dynamics of the riser.

In recent years, similar method has been used to simulate the blades of wind turbine considering deformation. Meng et al.[133] proposed a new elastic actuator line model, which is the combination of the actuator line (AL) wake model and a finite difference structural model. The coupled code was used to simulate a single NREL 5 MW turbine and the results showed a good agreement for both high and low TSRs. Ma et al.[134] proposed a actuator line finite-element beam method (ALFBM) which is similar in concept to the previous method. All the structural details are taken into account including the tilt angle of the main shaft and the precone angle of the rotor. Many characteristics are studied so as to demonstrate all the coupled aeroelastic wake behaviours of the wind turbine.

5. Fluid-noise coupling

Fluid induced noise is one of the most challenging problems for ships and underwater submarines. During these years, lots of numerical work has been done in order to predict the sound propagation. The effect of noise on the fluid is usually ignored due to its low energy. Thus the simulation can be handled as a one-way coupling process. At present, the calculation of flow-induced noise is mainly based on the acoustic analogy, which includes Lighthill equations, Curler integral solutions and FW-H equations. For the implementation of numerical simulation, Green's function is the most commonly used method to calculate the sound pressure distribution, while there are some studies using the finite volume method for more accurate results. Application softwares also range from commercial FLUENT software to open source OpenFOAM and CALFEM.

From the perspective of application software, scholars are gradually shifting from commercial software to open source software. Toth et al.[135] used two software, FLUENT and OpenFOAM, to simulate the two-dimensional flow sound pressure propagation process, and found that mesh refinement can significantly reduce calculation errors. And it was found that the density-based algorithm is better than the pressure-based solution. Nillson[136] combined OpenFOAM and CALFEM to carry out fluid induced noise simulation. Incompressible flows data solved by OpenFOAM were adopted to construct sound source, which was then input into CALFEM to solve the noise based on Finite Element Method. Open cavity flow was adopted as typical case in the study. Schmalz and Kowalczyk[137] used the Curle integral solution to simulate the turbulence-based sound source and sound pressure wave propagation process. With the help of acousticFoam, a modified application of OpenFOAM is developed to compute and solve incompressible fluid dynamic simulation cases regarding turbulence inducted noise. The quadratic brace 2-D-simulation case was run in parallel on 8 processors, which was a parallelized computation case. Bensow and Liefvendahl[138] used a propagation modelling methodology based on the Ffowcs Williams and Hawkings approach. Green function of Francescantonio format was adopted based on the solver in OpenFOAM. The model used catamarans and small scientific research ships, and the influence of the hull is considered when calculating the propeller noise. However, the free surface is neglected in the simulation. Based on the LES and FW-H equations, Cianferra et al.[139] converted the quadrupoles in the sound source into the form of convection terms and directly calculated the volume integrals. In a uniform flow with a Reynolds number of 4330, the turbulent noise in three geometric shapes, sphere, square, and ellipsoid, were calculated. The results showed that for a bluff body, the loading noise, or in other words, dipole noise is much larger than the quadrupole noise, while the streamlined body is the opposite. The results showed that the persistence of a two-dimensionally shaped wake when compared to a 3-D one contributes to increasing the quadrupole part of the radiated noise.

Standard model tests are often used to verify the accuracy of a calculation method. Lanniello et al.[140] used different forms of RANS and FW-H solutions to calculate steady propeller problems. The numerical simulation results were compared with the experimental results under three different advance coefficients. It was found that the linear term of FW-H only plays a dominant role in the space range closer to the propeller, and the nonlinear term plays a dominant role in the far-field region. However, the cavitation of the propeller is not considered. As Fig. 15 shows, the test positions are arranged along the propeller axis direction. The left picture shows the sketch of the calculation using the quadrupole sound source, and the right picture shows the cylindrical control surface selected in which the quadrupole sources were generated.

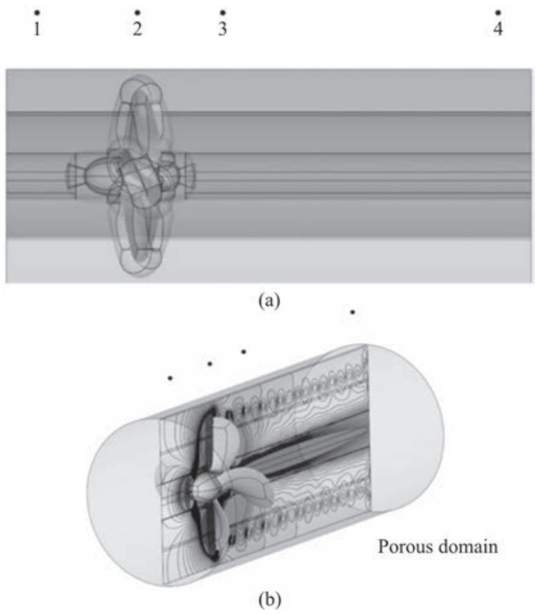

Fig. 15 The hydrophones position with respect to the 3-D meshes used to evaluate the quadrupole sources (a), and the cylindrical surface used as the porous domain (b)[140]

Nitzkorski and Mahesh[141] used dynamic end cap methodology to add spurious contribution to the far field within the context of the Ffowcs-Williams and Hawkings. Combined with the FW-H equation, noise induced by flow passing a cylinder at the Reynolds numbers of 150, 10^4 and 8.9×10^4 were simulated. The reliability of the DNS method was first validated by the comparison with the experimental results at low Reynolds number. When the Reynolds number reaches 8.9×10^4, LES method was used. The proposed approach performed better than other commonly used approaches with the added benefit of computational efficiency and ability to query independent volumes. Lidtke et al.[142] used LES, Sauer and Schnerr mass conversion models and FW-H equation to simulate the flaky cavities produced by NACA0009 hydrofoil. The radiation noise is related to the cavitation volume and lift coefficient. This simulation method can accurately simulate the monopole noise caused by the vibrating sheet-shaped cavities. But when the cavitation bubble collapses into a small cavitation cloud, this numerical method cannot solve it well. As a supplement, Lidtke et al.[143] used FW-H acoustic analogy to calculate the blade passage noise, which is the noise generated by the passage of a rotor blade past a fixed object. The predicted results are in good agreement with the experiments. However, due to the use of URANS, the fluid instability is almost zero, which is not suitable for high-precision bubble dynamics and shock structure. Zhang et al.[144] calculated the noise generated by a NACA0012 airfoil in the wake of a rod with two kinds of methods. One is the LES simulation combined with FW-H acoustic analogy, the other is LES simulation combined with Powell vortex sound theory. Both of them were proved to be robust, credible and satisfactory.

Facing the actual needs of engineering, there is generally stronger needs to simulate some more practical examples. By using the potential flow theory, Testa[145] compared the unsteady Bernoulli equation and FW-H equation in the prediction of cavitation noise under open-water conditions. Transpiration rate model (TVM) was used to model the cavitation. Ianniello et al.[146] predicted the underwater noise of full-scale ship models in steady flow using acoustic analogy. The level set approach was used to take into account the time evolution of the free surface. It was found that due to the influence of the hull surface, the noise scattering effect was obvious, and sound waves may be reflected by the free surface. Besides, the turbulent pulsation component in the velocity field still played a major role. On the other hand, the calculated pressure field time history was converted into a frequency domain map and into decibel units. The results showed that the acoustic analogy method can predict well in both the direction and amplitude of noise propagation. Lloyd et al.[147] also used the acoustic analogy method based on the results of hydrodynamic simulation to perform noise prediction for a rotating propeller. Compared with the acoustic analogy method used in far field, grids were designed to improve the resolution of the pressure field away from the propeller. The study found that the local pressure at a field point was shown to be more sensitive to grid refinement than integrated propeller performance coefficients. In the far field, the pressure amplitude was small, and the error effect of numerical simulation increased, especially when studying high-frequency noise. Therefore, when using acoustic simulation methods, accurate hydrodynamic convergence solutions are important. Ianniello[148] applied the FW-H equation to the acoustic analysis and explored the conditions where the non-linear noise sources of the blade can be ignored. The paper adopted an empirical method (using a rotating point source, named “rotpole” here), and used its surface integral kernels to replace the multi-leaf rotating structure. The results showed that underwater noise prediction is an inherently nonlinear problem and, contrary to analogous aeronautical configurations, it always requires an accurate estimation of the nonlinear flow sources just by virtue of the very low rotational speed. Cianferra et al.[149] used the FW-H formulation to validate time delays analysis in the case of an acoustic monopole field. A finite-length cylinder with square section was simulated through turbulent large eddy simulation. A scale effect analysis was performed in the monopole noise domain involving source length, maximum frequency and speed of sound, and a criterion based on the value of the non-dimensional parameter maximum frequency parameter (MFP) was given.

6. Future directions

With the rapid development of computer technology, numerical simulation methods should be developed for more efficient simulation of more complex and real situations in the future. The future directions can be summarized as the following four points:

(1) The coupling of more complex and extreme fully real ocean conditions should be realized in numerical simulation to 100% reproduce the environmental conditions of ships and marine engineering structures in actual working conditions.

(2) In the future, multi-physical field solution under multiple systems should be realized simultaneously in simulation, and comprehensive performance prediction can be carried out at the same time. Especially, polar ships are becoming the focus of ocean transportation and resource development. Low temperature environment and floating ice have put forward higher requirements for the com- prehensive performance of ships. At the same time, numerical simulation methods for evaluating ship performance in polar region are also in urgent need of development.

(3) For some special flows with large spatial scale and complex temporal evolution, it is necessary to conduct full-scale time-space coupling simulation to fully capture the flow characteristics.

(4) In order to meet the requirements of highly coupling simulation in the future, it is necessary to develop high performance parallel coupling simulation software suitable for supercomputing platform. Adapted hardware develops from traditional CPU to GPU. The scale of computation develops from hundred/thousand cores to ten thousand cores or even a million cores.

7. Conclusion

Numerical simulation has played an important role in solving the coupling hydrodynamic problems in ship and ocean engineering field. On the one hand, many numerical solving techniques were proposed in order to predict the coupled effect quickly, such as body force model, boundary element analysis, actuator line model and strip method. On the other hand, many numerical solving techniques were proposed in order to capture the detailed coupled flow information, such as sliding grid method, overset grid method and meshless-FEM coupling method. By a serious of verification and validation, the applicability and reliability of most numerical simulation techniques have been proved to be acceptable in the evolution and design for ship and ocean engineering structures. Taking advantage of the proposed practical models, many complex coupling problems in ship and ocean engineering field have been solved by CFD techniques. Simulation based design and optimization have become an important trend in the modern ship and ocean engineering industry. In the future, numerical simulation will be developed to predict the comprehensive performance under larger scale and more complex conditions, and play a greater role in the design of ships and marine engineering structures.

Acknowledgements

This work was supported by the Chang Jiang Scholars Program (Grant No. T2014099), the Shanghai Excellent Academic Leaders Program (Grant No. 17XD1402300) and the Innovative Special Project of Numerical Tank of Ministry of Industry and Information Technology of China (2016-23/09).

References

[1] Stern F., Yang J., Wang Z. et al. Computational ship hydrodynamics: Nowadays and way forward [J]. *International Shipbuilding Progress*, 2013, 60: 3-105.

[2] Stern F., Kim H. T., Patel V. C. et al. A viscous-flow approach to the computation of propeller-hull interaction [J]. *Journal of Ship Research*, 1988, 32(4): 246-262.

[3] Kawamura T., Miyata H., Mashimo K. Numerical simulation of the flow about self-propelling tanker models [J]. *Journal of Marine Science and Technology*, 1997, 2(4): 245-256.

[4] Choi E., Kim J. H., Lee H. G. et al. Computational predictions of ship-speed performance [J]. *Journal of Marine Science and Technology*, 2009, 14(3): 322-333.

[5] Choi J. E., Min K. S., Kim J. H. et al. Resistance and propulsion characteristics of various commercial ships based on CFD results [J]. *Ocean Engineering*, 2010, 37(7): 549-566.

[6] Phillips A. B., Turnock S. R., Furlong M. Accurate capture of propeller-rudder interaction using a coupled blade element momentum-RANS approach [J]. *Ship Technology Research*, 2010, 57(2): 128-139.

[7] Simonsen C. D., Stern F. RANS maneuvering simulation of Esso Osaka with rudder and a body-force propeller [J]. *Journal of Ship Research*, 2005, 49(2): 98-120.

[8] Dubbioso G., Durante D., Broglia R. Zig-zag maneuver simulation by CFD for tanker like vessel [C]. *Proceedings of the 5th International Conference on Computational Methods in Marine Engineering*, Hamburg, Germany, 2013, 29-31.

[9] Dubbioso G., Durante D., Di Mascio A. et al. Turning ability analysis of a fully appended twin screw vessel by CFD. Part II: Single vs. twin rudder configuration [J]. *Ocean Engineering*, 2016, 117: 259-271.

[10] Mofidi A., Martin J. E., Carrica P. M. Propeller/rudder interaction with direct and coupled CFD/potential flow propeller approaches, and application to a zigzag manoeuvre [J]. *Ship Technology Research*, 2018, 65(1): 10-31.

[11] Kaidi S., Smaoui H., Sergent P. Numerical estimation of bank-propeller-hull interaction effect on ship manoeuvring using CFD method [J]. *Journal of Hydrodynamics*, 2017, 29(1): 154-167.

[12] Lübke L. O. Numerical simulation of the flow around the propelled KCS [C]. *Proceedings of CFD Workshop 2005*, Tokyo, Japan, 2005, 9-11.

[13] Queutey P., Deng G., Wackers J. et al. Sliding grids and adaptive grid refinement for RANS Simulation of ship-propeller interaction [J]. *Ship Technology Research*, 2012, 59(2): 44-57.

[14] Zhu Q., Wang Z., Ling H. et al. Use of the sliding mesh technique to forecast ship self-propulsion performance with propeller in open water [J]. *Ship Science and Technology*, 2016, 38(9): 42-48.

[15] Badoe C. E., Phillips A. B., Turnock S. R. Influence of drift angle on the computation of hull–propeller–rudder interaction [J]. *Ocean Engineering*, 2015, 103: 64-77.

[16] Moctar O., Lantermann U., Mucha P. et al. RANS-based simulated ship maneuvering accounting for hull-propulsor-engine interaction [J]. *Ship Technology Research*, 2014, 61(3): 142-161.

[17] Paik K. J., Park H. G., Seo J. URANS simulations of cavitation and hull pressure fluctuation for marine propeller with hull interaction [C]. *Third International Symposium on Marine Propulsors*, Launceston, Tasmania, Australia, 2013..

[18] Fujiyama K. Numerical simulation of ship hull pressure fluctuation induced by cavitation on propeller with capturing the tip vortex [C]. *Fourth International Symposium on Marine Propulsors*, Austin, Texas, USA, 2015.

[19] Han C. Z., Long Y., Ji B. et al. An integral calculation approach for numerical simulation of cavitating flow around a marine propeller behind the ship hull [J]. *Journal of Hydrodynamics*, 2018, 30(6): 1186-1189.

[20] Long Y., Long X., Ji B. et al. Numerical simulations of cavitating turbulent flow around a marine propeller behind the hull with analyses of the vorticity distribution and particle tracks [J]. *Ocean Engineering*, 2019, 189: 106310.

[21] Liu D. C., Zhou W. X. Numerical predictions of the propeller cavitation pressure fluctuation behind ship and comparison with experiment [J]. *Journal of Ship Mechanics*, 2019, 23(3): 245-254(in Chinese).

[22] Carrica P. M., Ismail F., Hyman M. et al. Turn and zigzag maneuvers of a surface combatant using a URANS approach with dynamic overset grids [J]. *Journal of Marine Science and Technology*, 2013, 18(2): 166-181.

[23] Mofidi A., Carrica P. M. Simulations of zigzag maneuvers for a container ship with direct moving rudder and propeller [J]. *Computers and Fluids*, 2014, 96: 191-203.

[24] Shen Z., Wan D., Carrica P. M. Dynamic overset grids in OpenFOAM with application to KCS self-propulsion and maneuvering [J]. *Ocean Engineering*, 2015, 108: 287-306.

[25] Meng Q. J., Wan D. C. Numerical simulations of viscous flow around the obliquely towed KVLCC2M model in deep and shallow water [J]. *Journal of Hydrodynamics*, 2016, 28(3): 506-518.

[26] Wang J., Zhao W., Wan D. Free maneuvering simulation of ONR tumblehome using overset grid method in naoe-FOAM-SJTU solver [C]. *Proceedings of 31th Symposium on Naval Hydrodynamics*, Monterey, USA, 2016.

[27] Wang J., Liu X., Wan D. et al. Numerical prediction of KCS Self-propulsion in shallow water [C]. *Proceedings of the Twenty-sixth International Ocean and Polar Engineering Conference*, Rhodes, Greece, 2016.

[28] Wang J. H., Wan D. C. Investigation of self-propulsion in waves of fully appended ONR tumblehome model [J]. *Applied Mathematics and Mechanics (English Edition)*, 2016, 37(12): 1345-1358.

[29] Wang J., Zou L., Wan D. CFD simulations of free running ship under course keeping control [J]. *Ocean Engineering*, 2017, 141: 450-464.

[30] Wang J. H., Wan D. C. CFD Investigations of ship maneuvering in waves using naoe-FOAM-SJTU solver [J]. *Journal of Marine Science and Application*, 2018, 17(3): 443-458.

[31] Lu L., Cheng L., Teng B. et al. Numerical simulation and comparison of potential flow and viscous fluid models in near trapping of narrow gaps [J]. *Journal of Hydrodynamics*, 2010, 22(5 Suppl. 1): 120-125.

[32] Garrison C. J. Hydrodynamic loading of large offshore structures: Three-dimensional source distribution methods. Numerical methods in offshore engineering [M]. Nottingham, UK: A Wiley-Interscience Publication, 1978, 87-140.

[33] Teng B., Eatock T. R. New higher-order boundary element methods for wave diffraction/radiation [J]. *Applied Ocean Research*, 1995, 17: 71-78.

[34] Teng B., Bai W., Xiang Y. A B-spline based BEM and its application in predicting wave forces on 3D bodies [J]. *China Ocean Engineering*, 1999, 13(3): 257-264.

[35] Wang Q., Zhou W., Cheng Y. et al. A line integration method for the treatment of 3D domain integrals and accelerated by the fast multipole method in the BEM [J]. *Computational Mechanics*, 2016, 59(4):1-14.
[36] Zhang S., Li X. A self-adaptive projection method for contact problems with the BEM [J]. *Applied Mathematical Modelling*, 2018, 55: 145-159.
[37] Chau F. P., Taylor R. E. Second-order wave diffraction by a vertical cylinder [J]. *Journal of Fluid Mechanics*, 1992, 240: 571-599.
[38] Malenica S., Molin B. Third-harmonic wave diffraction by a vertical cylinder [J]. *Journal of Fluid Mechanics*, 1995, 302: 203-229.
[39] Isaacson M., Cheung K. F. Second order wave diffraction around two-dimensional bodies by time-domain method [J]. *Applied Ocean Research*, 1991, 13(4): 175-186.
[40] Duan W., Chen J., Zhao B. Second-order Taylor Expansion Boundary Element Method for the second-order wave diffraction problem [J]. *Engineering Analysis with Boundary Elements*, 2015, 58: 12-26.
[41] Xu G., Hamouda A. M. S., Khoo B. C. Time-domain simulation of second-order irregular wave diffraction based on hybrid water wave radiation condition [J]. *Applied Mathematical Modelling*, 2016, 40(7): 4451-4467.
[42] Longuet-Higgins M. S., Cokelet E. D. The deformation of steep surface waves on water. I. A numerical method of computation [J]. *Proceedings of the Royal Society A: Mathematical, Physical and Engineering Sciences*, 1976, 350(1660): 1-26.
[43] Grilli S. T., Subramanya R. Numerical modeling of wave breaking induced by fixed or moving boundaries [J]. *Computational Mechanics*, 1996, 17(6): 374-391.
[44] Stéphan T., Grilli G. P., Frédéric D. A fully non-linear model for three-dimensional overturning waves over an arbitrary bottom [J]. *International Journal for Numerical Methods in Fluids*, 2001, 35(7): 829-867.
[45] Feng A. Numerical simulation of nonlinear wave-body problem based on desingularized Rankine source and mixed Euler-Lagrange method [D]. Doctoral Thesis, Southampton, UK: University of Southampton, 2014.
[46] Zhou B. Z., Teng B., Chen L. et al. Modeling of fully nonlinear wave evolution over a submerged bar [J]. *Journal of Hydrodynamics*, 2010, 22(5 Suppl. 1): 83-90.
[47] Teng B., Gou Y. BEM for wave interaction with structures and low storage accelerated methods for large scale computation [J]. *Journal of Hydrodynamics*, 2017, 29(5): 748-762.
[48] Castiglione T., Stern F., Bova S. et al. Numerical investiation of the seakeeping behavior of a catamaran advancing in regular head waves [J]. *Ocean Engineering*, 2011, 38(16): 1806-1822.
[49] Shen Z. R., Ye H. X., Wan D. C. Motion response and added resistance of ship in head waves based on RANS simulations [J]. *Chinese Journal of Hydrodynamics*, 2012, 27(6): 621-633(in Chinese).
[50] Ye H. X., Shen Z. R., Wan D. C. Numerical prediction of added resistance and vertical ship motions in regular head waves [J]. *Journal of Marine Science and Application*, 2012, 11(4): 410-416.
[51] Shen Z. R., Ye H. X., Wan D. C. URANS simulations of ship motion responses in long-crest irregular waves [J]. *Journal of Hydrodynamics*, 2014, 26(3): 436-446.
[52] Liu C., Wang J., Wan D. CFD Computation of wave forces and motions of DTC ship in oblique waves [J]. *International Journal of Offshore and Polar Engineering*, 2018, 28(2): 154-163.
[53] Danmeier D. G., Seah R. K. M. Validation of wave run-up calculation methods for a gravity based structure [C]. *27th International Conference on Offshore Mechanics and Arctic Engineering*, Estoril, Portugal, 2008.
[54] Cao H. J., Wan D. C. RANS-VOF solver for solitary wave run-up on a circular cylinder [J]. *China Ocean Engineering*, 2015, 29(2): 183-196.
[55] Liu Y., Wan D. C. Numerical simulation of motion response of an offshore observation platform in waves [J]. *Journal of Marine Science and Application*, 2013, 12(1): 89-97.
[56] Liu Y., Peng, Y., Wan D. Numerical investigation on interaction between a semi-submersible platform and its mooring system [C]. *34th International Conference on Ocean, Offshore and Arctic Engineering*, St. John's, Newfoundland, Canada, 2015.
[57] Zhuang Y., Wan D. Numerical study of focused waves acting on a fixed FPSO-shaped body [J]. *International Journal of Offshore and Polar Engineering*, 2019, 29(2): 128-140.
[58] Carrica P. M., Wilson R. V. Unsteady RANS simulation of the ship forward speed diffraction problem [J]. *Computers and Fluids*, 2006, 35(6): 545-570.
[59] Carrica P. M., Wilson R. V. Ship motions using single-phase level set with dynamic overset grids [J]. *Computers and Fluids*, 2006, 36(9): 1415-1433.
[60] Bihs H., Kamath A., Chella M. A. et al. A new level set numerical wave tank with improved density interpolation for complex wave hydrodynamics [J]. *Computers and Fluids*, 2016, 140: 191-208.
[61] Bihs H., Kamath A., Chella M. A. et al. Breaking-wave interaction with tandem cylinders under different impact scenarios [J]. *Journal of Waterway, Port, Coastal, and Ocean Engineering*, 2016, 142(5): 04016005.
[62] Chella M. A., Bihs H., Myrhaug D. et al. Breaking solitary waves and breaking wave forces on a vertically mounted slender cylinder over an impermeable sloping seabed [J]. *Journal of Ocean Engineering and Marine Energy*, 2017, 3(1): 1-19.
[63] Teng B., Mao H. F., Ning D. Z. et al. Viscous numerical examination of hydrodynamic forces on a submerged horizontal circular cylinder undergoing forced oscillation [J]. *Journal of Hydrodynamics*, 2019, 31(5): 887-899.
[64] Khayyer A., Gotoh H., Shao S. D. Corrected incompressible SPH method for accurate water-surface tracking in breaking waves [J]. *Coastal Engineering*, 2008, 55(3): 236-250.
[65] Cleary P. W., Rudman M. Extreme wave interaction with a floating oil rig: Prediction using SPH [J]. *Progress in Computational Fluid Dynamics*, 2009, 9(6-7): 332-334.
[66] Rudman M., Cleary P. W. Rogue wave impact on a tension leg platform: The effect of wave incidence angle and mooring line tension [J]. *Ocean Engineering*, 2013, 61(6): 123-138.
[67] Rudman M., Cleary P. W. The influence of mooring system in rogue wave impact on an offshore platform [J]. *Ocean Engineering*, 2016, 115: 168-181.
[68] Ren B., Wen H., Dong P. et al. Improved SPH simulation of wave motions and turbulent flows through porous media [J]. *Coastal Engineering*, 2016, 107: 14-27.
[69] Shibata K., Koshizuka S., Sakai M. et al. Transparent boundary condition for simulating nonlinear water waves by a particle method [J]. *Ocean Engineering*, 2011, 38(16): 1839-1848.
[70] Zhang Y., Tang Z., Wan D. Numerical investigations of waves interacting with free rolling body by modified MPS

method [J]. *International Journal of Computational Methods*, 2016, 13(4): 1641013.
[71] Zhang Y., Wan D. Numerical study of interactions between waves and free rolling body by IMPS method [J]. *Computers and Fluids*, 2017, 155: 124-133.
[72] Rao C. P., Wan D. C. Numerical study of the wave-induced slamming force on the elastic plate based on MPS-FEM coupled method [J]. *Journal of Hydrodynamics*, 2018, 30(1): 70-78.
[73] Zhang G. Y., Chen X., Wan D. C. MPS-FEM coupled method for study of wave-structure interaction [J]. *Journal of Marine Science and Application*, 2019, 18(4): 387-399.
[74] Glauert H. Airplane propellers (Aerodynamic theory) [M]. Berlin, Germany: Springer,1928, 169-360.
[75] Kim S., Sclavounos P. D. Fully coupled response simulations of theme offshore structures in water depths of up to 10,000 feet [C]. *Proceedings of the Eleventh International Offshore and Polar Engineering Conference*, Stavanger, Norway: The International Society of Offshore and Polar Engineers, 2001.
[76] Kim S. LINES 2001: Nonlinear static and dynamic analysis of mooring line/riser/tether arrays, user manual [Z]. Cambridge, Massachusetts, USA: Massachusetts Institute of Technology, 2004.
[77] Kim S. MOTION 2001: Time-domain response analysis of offshore platforms, user manual [Z]. Cambridge, Massachusetts, USA: Massachusetts Institute of Technology, 2004.
[78] Kim S. SWIM 2001: Frequency-domain analysis of offshore platforms, user manual [Z]. Cambridge, Massachusetts, USA: Massachusetts Institute of Technology, 2004.
[79] Nielsen F. G., Hanson T. D., Skaare B. Integrated dynamic analysis of floating offshore wind turbines [J]. *American Society of Mechanical Engineers*, 2006, 1: 671-679.
[80] Jonkman J. M. Dynamics modeling and loads analysis of an offshore floating wind turbine [M]. Ann Arbor, Michigan, USA: ProQuest, 2007.
[81] Bae Y. H., Kim M. H., Shin Y. S. Rotor-floater-mooring coupled dynamic analysis of mini TLP-type offshore floating wind turbines [C]. *The 29th International Conference on Ocean, Offshore and Arctic Engineering*, Shanghai, China, 2010, 3: 491-498.
[82] Bachynski E. E., Kvittem M. I., Luan C. et al. Wind-wave misalignment effects on floating wind turbines: Motions and tower load effects [J]. *Journal of Offshore Mechanics and Arctic Engineering*, 2014, 136(4): 041902.
[83] Ma Y., Hu Z. Q., Xiao L. F. Wind-wave induced dynamic response analysis for motions and mooring loads of a spar-type offshore floating wind turbine [J]. *Journal of Hydrodynamics*, 2015, 26(6): 865-874.
[84] Shen M., Hu Z., Geng T. Coupled hydrodynamic and aerodynamic response analysis of a tension-leg platform floating wind turbine [J]. *Journal of Ship Mechanics*, 2017, 21(3): 263-174.
[85] Zhou T., He Y., Meng L. Dynamic response analysis of a 6 MW spar-type floating offshore wind turbine under second-order wave forces [J]. *Journal of Harbin Institute of Technology*, 2018, 50(4): 145-152.
[86] Qu X., Tang Y., Li Y. et al. Motion performance analysis of a floating offshore wind turbine with single-point mooring system under misaligned wind and wave condition [J]. *Journal of Harbin Institute of Technology*, 2018, 39(8): 1328-1336.
[87] Guo Y., Liu L. Q., Li Y. et al. The surge-heave-pitch coupling motions of the Φ type vertical axis wind turbine supported by the truss Spar floating foundation [J]. *Journal of Hydrodynamics*, 2019, 31(4): 669-681.
[88] Ren N., Li Y., Ou J. Coupled wind-wave time domain analysis of floating offshore wind turbine based on computational fluid dynamics method [J]. *Journal of Renewable and Sustainable Energy*, 2014, 6(2): 023106.
[89] Liu Y., Xiao Q., Incecik A. et al. Investigation of the effects of platform motion on the aerodynamics of a floating offshore wind turbine [J]. *Journal of Hydrodynamics*, 2016, 28(1): 95-101.
[90] Liu Y., Xiao Q., Incecik A. et al. Establishing a fully coupled CFD analysis tool for floating offshore wind turbines [J]. *Renewable Energy*, 2017, 112: 280-301.
[91] Quallen S., Xing T., Carrica P. et al. CFD simulation of a floating offshore wind turbine system using a quasi-static crowfoot mooring-line model [J]. *Journal of Ocean and Wind Energy*, 2014, 1(3): 143-152.
[92] Quallen S., Xing T. CFD simulation of a floating offshore wind turbine system using a variable-speed generator-torque controller [J]. *Renewable Energy*, 2016, 97: 230-242.
[93] Tran T., Kim D. Fully coupled aero-hydrodynamic analysis of a semi- submersible FOWT using a dynamic fluid body interaction approach [J]. *Renewable Energy*, 2016, 92: 244-261.
[94] Tran T. T., Kim D. H. A CFD study of coupled aerodynamic-hydrodynamic loads on a semisubmersible floating offshore wind turbine [J]. *Wind Energy*, 2018: 21(1): 70-85.
[95] Cheng P., Wan D. Fully Coupled aero-hydrodynamic simulation of floating offshore wind turbines with overset grid technology [C]. *Proceedings of the Fourth International Conference in Ocean Engineering*, Chennai, India, 2018.
[96] Lu H., Porté-Agel F. Large-eddy simulation of a very large wind farm in a stable atmospheric boundary layer [J]. *Physics of Fluids*, 2011, 23(6): 065101..
[97] Cheng P., Wan D. Fully-coupled aero-hydrodynamic simulation of floation offshore wind turbines by different simulation methods [C]. *Proceedings of the ASME 2018 37th International Conference on Ocean, Offshore and Arctic Engineering*, Madrid, Spain, 2018.
[98] Huang Y., Wan D., Hu C. Coupled aero-hydrodynamic analysis on a floating offshore wind turbine under extreme sea conditions [C]. *Proceedings of the Twenty-seventh (2017) International Ocean and Polar Engineering Conference*, San Francisco, CA, USA, 2017.
[99] Huang Y., Cheng P., Wan D. C. Numerical analysis of a floating offshore wind turbine by coupled aero-hydrodynamic simulation [J]. *Journal of Marine Science and Application*, 2019, 18(1): 82-92.
[100] Cheng P., Huang Y., Wan D. A numerical model for fully coupled aero-hydrodynamic analysis of floating offshore wind turbine [J]. *Ocean Engineering*, 2019, 173: 183-196.
[101] Ning X., Wan D. LES study of wake meandering in different atmospheric stabilities and its effects on wind turbine aerodynamics [J] . *Sustainability*, 2019, 11(24): 6939.
[102] Huang Y., Wan D. Investigation of interference effects between wind turbine and spar-type floating platform under combined wind-wave excitation [J]. *Sustainability*, 2020, 12(1): 246.
[103] Oberhagemann J., Holtmann M., Moctar O. et al. Stern slamming of a LNG carrier [J]. *Journal of Offshore Mechanics and Arctic Engineering*, 2009, 131(3):

031103.
[104] Maki K. J., Lee D., Troesch A. W. et al. Hydroelastic impact of a wedge-shaped body [J]. *Ocean Engineering*, 2011, 38(4): 621-629.
[105] Piro D. J, Maki K. J. Hydroelastic analysis of bodies that enter and exit water [J]. *Journal of Fluids and Structures*, 2013, 37: 134-150.
[106] Lakshmynarayanana P., Temarel P., Chen Z. Coupled fluid structure interaction to model three-dimensional dynamic behaviour of ship in waves [C]. *Proceedings of the 7th International Conference on Hydrodynamic in Marine Technology*, Split, Croatia, 2015.
[107] Qin H., Tang W., Xue H. et al. Dynamic response of a horizontal plate dropping onto nonlinear freak waves using a fluid–structure interaction method [J]. *Journal of Fluids and Structures*, 2017, 74: 291-305.
[108] Vuyst T. D., Vignjevic R., Campbell J. C. Coupling between meshless and finite element methods [J]. *International Journal of Impact Engineering*, 2005, 31: 1054-1064.
[109] Antoci C., Gallati M., Sibilla S. Numerical simulation of fluid-structure interaction by SPH [J]. *Computers and Structures*, 2007, 85(11-14): 879-890.
[110] Yang Q., Jones V., Mccue L. Free-surface flow interactions with deformable structures using an SPH–FEM model [J]. *Ocean Engineering*, 2012, 55(15): 136-147.
[111] Hu D., Long T., Xiao Y. et al. Fluid-structure interaction analysis by coupled FE-SPH model based on a novel searching algorithm [J]. *Computer Methods in Applied Mechanics and Engineering*, 2014, 276: 266-286.
[112] Long T., Hu D., Yang G. et al. A particle-element contact algorithm incorporated into the coupling methods of FEM-ISPH and FEM-WCSPH for FSI problems [J]. *Ocean Engineering*, 2016, 123: 154-163.
[113] Long T., Hu D., Wan D. et al. An arbitrary boundary with ghost particles incorporated in coupled FEM-SPH model for FSI problems [J]. *Journal of Computational Physics*, 2017, 350: 166-183.
[114] Lee C. J. K., Noguchi H., Koshizuka S. Fluid-shell structure interaction analysis by coupled particle and finite element method [J]. *Computers and Structures*, 2007, 85(11-14): 688-697.
[115] Mitsume N., Yoshimura S., Murotani K. et al. Improved MPS-FE fluid-structure interaction coupled method with MPS polygon wall boundary model [J]. *Computer Modeling in Engineering and Sciences*, 2014, 101(4): 229-247.
[116] Hwang S. C., Park J. C., Gotoh H. et al. Numerical simulations of sloshing flows with elastic baffles by using a particle-based fluid-structure interaction analysis method [J]. *Ocean Engineering*, 2016, 118: 227-241.
[117] Zhang Y. L., Chen X., Wan D. C. An MPS-FEM coupled method for the comparison study of liquid sloshing flows interacting with rigid and elastic baffles [J]. *Applied Mathematics and Mechanics (Engilsh Edition)*, 2016, 37(12): 1359-1377.
[118] Zhang Y., Wan D. MPS-FEM coupled method for sloshing flows in an elastic tank [J]. *Ocean Engineering*, 2018, 152: 416-427.
[119] Zhang Y., Wan D. MPS-FEM coupled method for fluid–structure interaction in 3D dam-break flows [J]. *International Journal of Computational Methods*, 2019, 16(2): 879-890.
[120] Chen X., Zhang Y., Wan D. Numerical study of 3-D liquid sloshing in an elastic tank by MPS-FEM coupled method [J]. *Journal of Ship Research*, 2019, 63(3): 143-153.
[121] Sun Z., Zhang G. Y., Zong Z. et al. Numerical analysis of violent hydroelastic problems based on a mixed MPS-mode superposition method [J]. *Ocean Engineering*, 2019, 179: 285-297.
[122] Han L., Hu X. SPH modeling of fluid-structure interaction [J]. *Journal of Hydrodynamics*, 2018, 30(1): 62-69.
[123] Khayyer A., Gotoh H., Falahaty H. et al. Towards development of enhanced fully-Lagrangian mesh-free computational methods for fluid-structure interaction [J]. *Journal of Hydrodynamics*, 2018, 30(1): 49-61.
[124] Zhang A. M., Sun P. N., Ming F. R. Smoothed particle hydrodynamics and its applications in fluid-structure interactions [J]. *Journal of Hydrodynamics*, 2017, 29(2): 5-34.
[125] Herfjord K., Drange S. O., Kvamsdal T. Assessment of vortex-induced vibrations on deepwater risers by considering fluid-structure interaction [J]. *Journal of Offshore Mechanics and Arctic Engineering*, 1999, 121(4): 207-212.
[126] Yamamoto C. T., Meneghini J. R., Saltara F. et al. Numerical simulations of vortex-induced vibration on flexible cylinders [J]. *Journal of Fluids and Structures*, 2004, 19(4): 467-489.
[127] Duanmu Y., Zou L., Wan D. C. Numerical simulations of vortex-induced vibrations of a flexible riser with different aspect ratios in uniform and shear currents [J]. *Journal of Hydrodynamics*, 2017, 29(6): 1010-1022.
[128] Duanmu Y., Zou L., Wan D. C. Numerical analysis of multi-modal vibrations of a vertical riser in step currents [J]. *Ocean Engineering*, 2018, 152: 428-442.
[129] Fu B. W., Zou L., Wan D. C. Numerical study on the effect of current profiles on vortex-induced vibrations in a top-tension riser [J]. *Journal of Marine Science and Application*, 2017, 16(4): 473-479.
[130] Fu B., Zou L., Wan D. Numerical study of vortex-induced vibrations of a flexible cylinder in an oscillatory flow [J]. *Journal of Fluids and Structure*, 2018, 77: 170-181.
[131] Bao Y., Palocios R., Sherwin S. Generalized thick strip modelling for vortex-induced vibration of long flexible cylinders [J]. *Journal of Computational Physics*, 2016, 321: 1079-1097.
[132] Bao Y., Zhu H. B., Huan P. et al. Numerical prediction of vortex-induced vibration of flexible riser with thick strip method [J]. *Journal of Fluids and Structures*, 2019, 89: 166-173.
[133] Meng H., Lien F. S., Li L. Elastic actuator line modelling for wake-induced fatigue analysis of horizontal axis wind turbine blade [J]. *Renewable Energy*, 2018, 116: 423-437..
[134] Ma Z., Zeng P., Lei L. P. Analysis of the coupled aero elastic wake behaviour of wind turbine [J]. *Journal of Fluids and Structures*, 2019, 84: 466-484.
[135] Tóth P., Fritzsch A., Lohász M. Application of computational fluid dynamics softwares for 2D acoustical wave propagation [C]. *Proceedings of Gépészet 2008 Conference*, Budapest, *Hungary*, 2008.
[136] Nilsson J. Implementation of acoustical analogies in openfoam and calfem [D]. Master Thesis, Lund, Sweden: University of Lund, 2010.
[137] Schmalz J., Kowalczyk W. Implementation of acoustic analogies in OpenFOAM for computation of sound fields [J]. *Open Journal of Acoustics*, 2015, 5(2): 56914.
[138] Bensow R., Liefvendahl M. An acoustic analogy and scale-resolving flow simulation methodology for the

prediction of propeller radiated noise [C]. *31th Symposium on Naval Hydrodynamics*, California, 2016.
[139] Cianferra M., Armenio V., Ianniello S. Hydroacoustic noise from different geometries [J]. *International Journal of Heat and Fluid Flow*, 2018, 70: 348-362.
[140] Ianniello S., Muscari R., Mascio A. Ship underwater noise assessment by the acoustic analogy. Part I: nonlinear analysis of a marine propeller in a uniform flow [J]. *Journal of marine science and technology*, 2013, 18(4): 547-570.
[141] Nitzkorski Z., Mahesh K. A dynamic end cap technique for sound computation using the Ffowcs Williams and Hawkings equations [J]. *Physics of Fluids*, 2014, 26(11): 115101.
[142] Lidtke A. K., Turnock S. R., Humphrey V. F. Characterisation of sheet cavity noise of a hydrofoil using the Ffowcs Williams–Hawkings acoustic analogy [J]. *Computers and Fluids*, 2016, 130: 8-23.
[143] Lidtke A. K., Humphrey V. F., Turnock S. R. Feasibility study into a computational approach for marine propeller noise and cavitation modelling [J]. *Ocean Engineering*, 2016, 120: 152-159.
[144] Zhang N., Xie H., Wang X. et al. Computation of vortical flow and flow induced noise by large eddy simulation with FW-H acoustic analogy and Powell vortex sound theory [J]. *Journal of Hydrodynamics*, 2016, 28(2): 255-266.
[145] Testa C. Acoustic formulations for aeronautical and naval rotorcraft noise prediction based on the ffowcs williams and hawkings equation [D]. Doctoral Thesis, Delft, The Netherlands: Technische Universiteit Delft, 2008.
[146] Ianniello S., Muscari R., Mascio A. Ship underwater noise assessment by the Acoustic Analogy part II: Hydroacoustic analysis of a ship scaled model [J]. *Journal of Marine Science and Technology*, 2014, 19(1): 52-74.
[147] Lloyd T., Rijpkema D, Van W. E. Marine propeller acoustic modelling: comparing CFD results with an acoustic analogy method [C]. *Proceedings of the Fourth International Symposium on Marine Propulsors (smp'15)*, Austin, TX, USA, 2015.
[148] Ianniello S. The Ffowcs Williams–Hawkings equation for hydroacoustic analysis of rotating blades. Part 1. The rotpole [J]. *Journal of Fluid Mechanics*, 2016, 797: 345-388.
[149] Cianferra M., Ianniello S., Armenio V. Assessment of methodologies for the solution of the Ffowcs Williams and Hawkings equation using LES of incompressible single-phase flow around a finite-size square cylinder [J]. *Journal of Sound and Vibration*, 2019, 453: 1-24.