Scholar Publishing Group
Frontiers in Ocean Engineering
https://doi.org/10.38007/FOE.2022.030401
ISSN 2790-2706 Vol. 3, Issue 4: 1-9

Hydrodynamic Numerical Calculation Method of Small-scale Objects in Ocean Engineering Based on Immersion Boundary Method

Shanshan Su*

Wuhan Donghu University, Wuhan, China

corresponding author

Keywords: Intrusion Boundary Method, Ocean Engineering, Small Scale Objects, Hydrodynamic Numerical

Abstract: Isolated piles, jacket platforms, submarine pipelines, etc. are ubiquitous structures in marine engineering. When the ratio of the lateral dimension D to the wavelength L of the part surrounded by waves is less than 0.2, it is generally called a small-scale structure. Current and waves are the two most important external loads in ocean engineering. The interaction between current and waves and small-scale structures in ocean engineering has always been the focus of research, and it is also one of the main problems that have not been well resolved in ocean engineering. In this paper, under the premise of considering the viscosity, turbulence and free surface flow of the fluid, the hydrodynamic problems related to small-scale objects in marine engineering are selected as the research content, and the numerical calculation model combining the immersion boundary method and the fluid volume method is selected for numerical calculation. The numerical expressions, solution steps and method verifications of the immersion boundary method and the fluid volume method are given respectively. A numerical calculation method by directly solving the external force source term in the immersion boundary method is proposed. The advantages, disadvantages and calculation steps of two different processing methods of applied force source terms in the immersion boundary method, continuous force method and discrete force method, are given. Instead of solving the force source term by means of interpolation and extrapolation, this paper adopts the discrete force method and the immersion boundary method for numerical calculation. Bounds method for numerical calculation. The realization process of establishing the numerical model of the immersion boundary method is given in detail. The finite difference method is used to discretize the governing equations and the semi-implicit two-step projection method is used to solve the NS equation. Finally, the numerical model of the immersion boundary method is given. Numerical realization process. The numerical model established in this paper is verified by the classical examples of numerical calculation of flow around a fixed cylinder and a flow around a rotating cylinder under laminar flow conditions, and the numerical calculation results are compared with the experimental and numerical results of others. The established numerical calculation model is correct and feasible. Experiments show that this model can well generate first-order Stokes linear waves. When a=0.25, according to the method proposed in this paper, the lift can be reduced to 75%-80%; when a=0.5, the lift can be reduced to 60%; when a=0.75, the lift amplitude can be suppressed to 50% or less.

1. Introduction

In a broad sense, the scope of marine engineering research includes all projects involving or related to the marine environment, such as offshore oil platforms, submarine oil pipelines, mooring systems and other structures that provide related production and living facilities for the development of marine resources and other activities. The technical research in marine engineering mainly includes deep-sea manned submersible technology, subsea oil pipeline burying and maintenance technology, marine space utilization and offshore construction technology. At present, the main research contents of ocean engineering include the laboratory research of ship and ocean engineering numerical pool simulation, the numerical simulation of the interaction between waves and structures, the numerical simulation of hydrodynamic problems in the deep sea, and the relationship between full-scale ships and marine structures. Massively Parallel Numerical Computing Simulation of Complex Flow Problems. In recent years, with the rapid development of marine engineering technology, the research on hydrodynamic problems around marine engineering structures has become more and more detailed. Current and waves are the two most important external loads in marine engineering. At the same time, the interaction between current and waves and marine engineering structures has always been the focus of research, and it is also one of the main problems that have not been well solved in marine engineering. [1-2] .

In the research on the hydrodynamic numerical calculation method of small-scale objects in marine engineering based on the immersion boundary method , many scholars have studied it and achieved good results, for example : Sawant VA et al. foundation of theoretical fluid mechanics. Theoretical analysis method is to establish basic equations or equation sets for various physical quantities in the fluid motion process, solve them in combination with specific calculation conditions, and analyze the calculation results reasonably [3] . Betaieb E et al. used some dummy particles to determine the free surface, and obtained the numerical solution of the large sloshing problem of the fluid with the free surface by using this method. The dummy particles are marked by the motion of the local fluid velocity, the area occupied by all fluids is defined as the fluid area, and the interface between the marked and non-marked areas is defined as the free surface [4] .

In this paper, considering the viscosity, turbulence and free surface flow of the fluid, the hydrodynamic problems related to small-scale objects in marine engineering are selected as the research content, and the numerical values of the immersed boundary method and the fluid volume method are given respectively. Expression, solution steps, and method validation. At the same time, the immersion boundary method and the fluid volume method are combined to carry out the verification calculation of relevant examples, and the numerical calculation of the near-wall cylinder under the action of waves is carried out by this method. In addition, this paper uses the open source numerical calculation software Openfoam to establish an H-dimensional numerical water tank model and verifies the correctness of the established numerical water tank. The problem of interaction between waves and structures is solved by using the established numerical water tank.

2. Research on the Numerical Calculation Method of Hydrodynamics of Small-Scale Objects in Marine Engineering Based on the Immersion Boundary Method

2.1. Continuous Force Method and Discrete Force Method

The calculation of each time step in the time discrete process of the continuous force method is basically the same, and the calculation at each time step can be summarized as:

(1) Calculate the magnitude of the external force according to the position and mechanical properties of the discrete points on the solid boundary;

(2) Calculate the force of the surrounding grid nodes by extrapolation according to the applied force calculated in step (1);

(3) Solve the NS equation in the entire flow field:

(4) According to the new velocity field, solve the new position of the discrete points of the solid boundary [5-6] .

The discrete force method obtains more accurate boundary conditions of the object surface by changing the numerical discrete method near the object surface boundary when processing the object surface boundary. Compared with the continuous force method, the discrete force method solves the force source terms or boundary conditions directly on the flow field nodes near the surface boundary of the object, instead of solving the force source terms through interpolation or extrapolation. Like the continuous force method, the discrete force method also has many different methods. There are many forms of governing equations, but the NS equation (momentum equation) will definitely add a new physical force source term, and some discrete force methods will also add mass to the continuity equation. source item. The additional force source terms and mass source terms can usually be obtained by solving the discrete governing equations. Similar to the continuous force method, the discrete force method differs in the time and space numerical discrete process according to the selected discrete method, but the calculation of each time step in the time discrete process is basically the same, and the The calculation can be summarized as:

(1) Solve the applied force at the current time step through discrete equations;

(2) Discretize the entire flow field, and use the two-step projection method to solve the intermediate velocity field of the first time step;

(3) Use the continuity equation to solve the pressure Poisson equation to obtain the pressure:

(4) Use the two-step projection method to solve the velocity field of the new time step [7-8] .

By comparing the numerical discrete processes of the two methods, the main differences between the continuous force method and the discrete force method are:

(1) The solution methods of the external force source term are different. The applied force source term of the continuum force method is solved in the coupled equation by integral form, usually before discretizing the master equation. The applied force source term of the discrete force method is solved by the discrete governing equation;

(2) The relationship between the external force source term and the spatial dispersion is different. Since the applied force source term of the continuous force method is solved before the discretization of the main governing equation, the applied force source term has nothing to do with the specific spatial discretization method. The applied force source term of the discrete force method has a great relationship with the spatial discrete method;

(3) The pressure value is calculated differently. In the continuous force method, the change of the flow field has nothing to do with the pressure value, but only with the pressure gradient. The discrete force method uses the pressure value when the two-step projection method is used to solve the calculation. Usually, the pressure field needs to be given in the initial conditions of the solution.

The continuous force method has a simple structure and strong practicability, but the discrete force method is higher than the continuous force method in the accuracy of the object surface boundary. The discrete force method is used in this paper .

2.2. Numerical Study of VOF

The volume-of-fluid method (VOF) does not capture the free surface by tracking the motion of the particles on the free surface, but determines the free surface by calculating the ratio function F between the fluid and the mesh volume in the mesh element. If F=1, it means that the mesh element is all fluid; if F=0, the mesh element is an empty mesh, which means air: when 0<F<1, the mesh element is a free surface mesh .

Assuming any point (x, y) in the two-dimensional flow field, the function f(x, y, t) is defined as follows [11-12] :

$$f(x,y,t)=\begin{cases}0 air\\1 water\\0\sim 1 freesurface\end{cases} \tag{1}$$

thus,

$$F=\frac{\int_{V_{i,j}} f(x,y,t)dV}{V_{i,j}} \tag{2}$$

where V i , j represent the volume of the computing unit grid (i, j).

2.3. Treatment Method of Free Surface Flow

The main feature of the free surface problem is that part of the boundary of the computational domain changes with time, and the boundary itself, together with the basic flow variables such as velocity and pressure, need to be solved simultaneously. Generally, the solution of differential equation system is assumed to be discretely solved in a fixed domain, and the calculation domain that changes itself undoubtedly greatly increases the complexity of the calculation, which is also the biggest difficulty of free surface flow.

The interface capture method is not as accurate as the tracking method on the free surface, but it can adapt to the complex interface flow problem. Since the snapping method is generally performed under a fixed Euler grid, the distortion and deformation of the grid can be avoided. The implementation process of the interface capture class method is generally complicated, and needs to be realized by means of special interface marking technology. So far people have developed a variety of interface capture methods [13-14] .

Generally speaking, the interface tracking method is simple to implement and has a small amount of computation, but it has great limitations in complex problems. The advantages of the interface capture method are mainly reflected in its strong adaptability to complex problems. It can handle almost all free surface problems, but the implementation process is relatively complex and the amount of calculation is large. In this paper, the capture method is used to calculate the free surface problem, which mainly considers that many hydrodynamic problems of marine engineering often involve complex forms such as large deformation and fragmentation of the free surface. In addition, interface capture methods are generally used in conjunction with one equation, which facilitates simultaneous consideration of turbulent phenomena in free surface flow [15-16] .

3. Research and Design Experiment of Hydrodynamic Numerical Calculation Method for Small-Scale Objects in Marine Engineering Based on Immersion Boundary Method

3.1. Research on the Establishment and Application of 3D Numerical Wave Flume

The essence of establishing a numerical water tank is to numerically realize the shape and position of the free surface at different times and the solution of the flow field in the area below the free surface. How to accurately simulate the free surface is the key to the establishment of a numerical water tank. This chapter will use the VOF method. Capture the shape and position of the free surface. At the same time, the open source numerical calculation software OpenFOAM will be used, the finite volume method will be used to solve the NS equation, the relaxation wave method proposed above will be used to create waves, and the wave problem will be solved by modifying the interFoam.C file under the existing solver interFoam in OpenFoam. The related solver waveFoam [17-18] .

3.2. Experimental Design

This paper , the interface reconstruction method of VOF and the basic idea of interface capture are discussed in detail, and the VOF equation is established. The time step selected in the numerical calculation must meet certain requirements. In order to maintain the stability of the numerical calculation, this chapter gives three constraints when using the VOF method to solve the problem. In order to test the reliability of the numerical model, this paper takes the complex free surface turbulent flow problem as an example to verify it numerically, mainly to verify the fluid sloshing problem and the laminar flow problem of dam-breaking flow when a two-dimensional rectangular tank is forced to move.

4. Experimental Analysis of the Hydrodynamic Numerical Calculation Method of Small-Scale Objects in Marine Engineering Based on the Immersion Boundary Method

4.1. Numerical Verification

In order to verify whether the numerical calculation method in this paper is correct, the newly established three-dimensional numerical wave tank model will be used to simulate the linear wave. This numerical wave pool is 20m long, 3m wide, 1m high, and has a water depth of h=0.65m. The positive X-axis is consistent with the wave propagation direction. The reflected wave often affects the results of the experiment, so the wave-elimination effect is very important for the numerical water tank simulation. In this example, the 3m at the exit is set as the wave-elimination area, as shown in Figure 1 . The adopted wave parameters are: wave height H=0.08m, period T=1.5s, wavelength 1=3.058m.

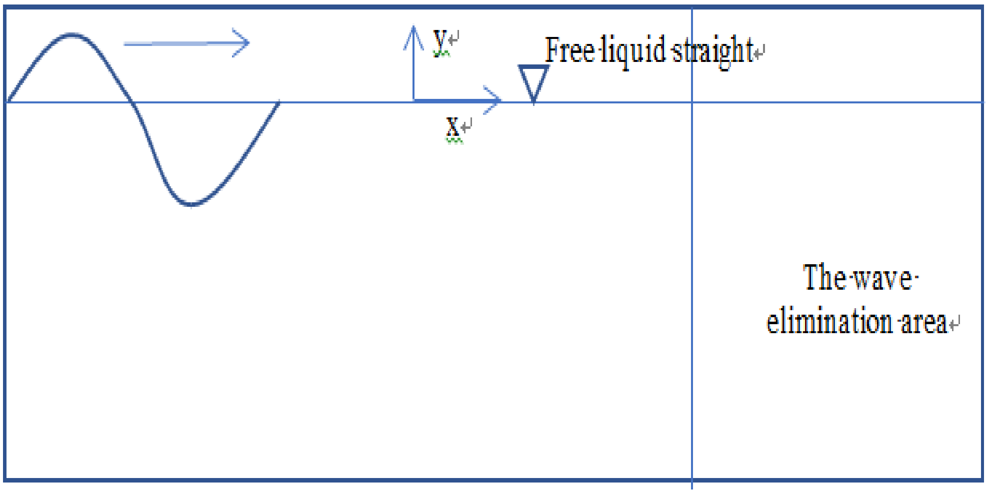

Figure 1. Schematic diagram of the numerical wave pool

x

Reasonable grid distribution is very important to generate an ideal wave field. In the numerical calculation of this paper, a uniform grid is used in the horizontal direction, and a non-uniform grid is used in the vertical direction. Encryption processing to obtain the free surface position more accurately. Table 1 presents the three grid parameters used in the numerical calculations.

Table 1. Numerically calculate the network parameters

reseau	△ x	△ z	△ y	Total number of grids
1	0.1	0.1	0.05	0.12
2	0.05	0.05	0.03	0.4
3	0.05	0.05	0.01	0.56

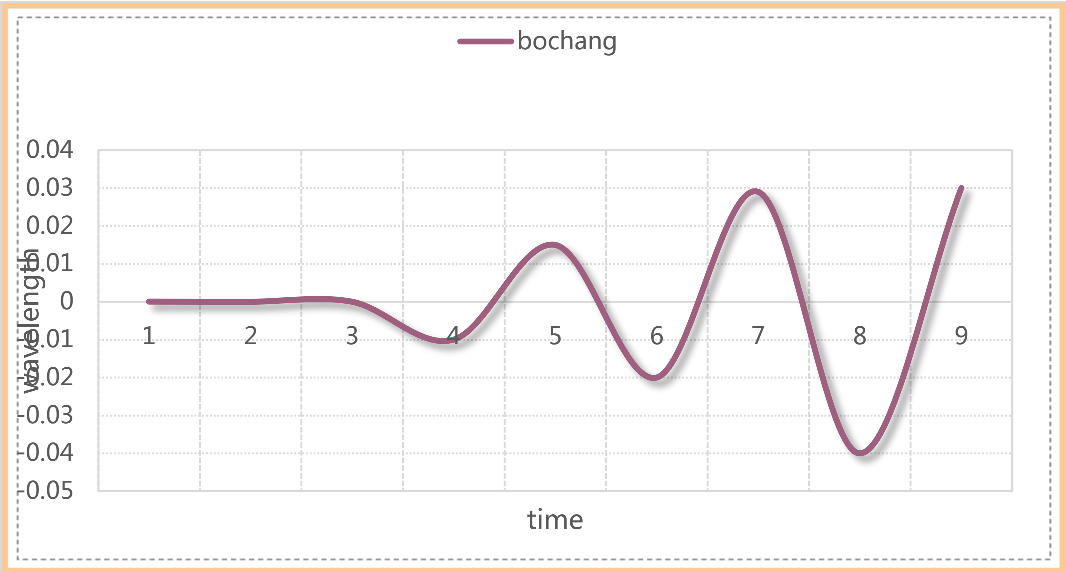

Figure 2. Displacement course curve at x=7.25m

The establishment of a numerical wave tank is to simulate the desired wave conditions, and an important parameter for evaluating and verifying whether a numerical wave tank is reasonable is the wave height factor . Figure 2 shows the displacement history curve at x=7.25m. It can be seen that the generated waveform is stable and the wave height is close to the theoretical wave height value, indicating that this model can well generate the first-order Stokes linear wave.

4.2. Comparison of Lift Control Efficiency under Different Reynolds Number and a Value Conditions

In this paper, experiments are designed for the lift control efficiency under different Reynolds numbers and values of a, and different Reynolds numbers and values of a are selected for comparison experiments of control variables.

Table 2. Comparison of lift control efficiency of different Reynolds numbers and a value conditions

Re	60	80	100	140	160	200
a=0.25	79	77	76	76	77	79
a=0.50	57	57	56	56	59	60
a=0.75	36	42	44	44	46	49

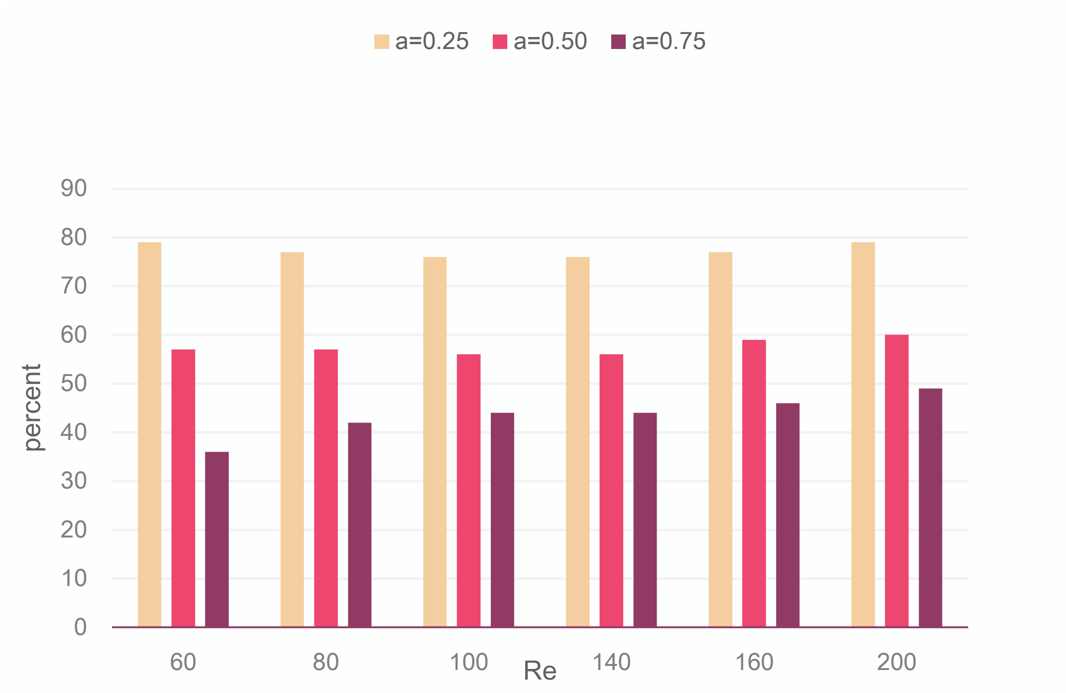

Figure 3. Changes in lift control efficiency at different Reynolds and a values

From Figure 3 : when a=0.25, the lift force can be reduced to 75%-80% according to the method proposed in this paper; when a=0.5, the lift force can be reduced to 60% level; when a=0.75 , the lift amplitude can be suppressed below 50%. This fully shows that the lift flow control scheme proposed in this paper can effectively reduce the lift on the cylinder, which is of great significance for solving the vortex-induced oscillation in engineering and the resulting fatigue damage.

5. Conclusion

This paper , the calculation steps and advantages and disadvantages of the discrete force method and the continuous force method, which are often used in numerical calculation, are given. The force source term and boundary conditions are solved on the flow field nodes, and the force source is not solved by means of interpolation and extrapolation. In this chapter, the discrete force method and the immersion boundary method are used for numerical calculation. The realization process of establishing the numerical model of the immersion boundary method is given in detail. The finite difference method is used to discretize the governing equation and the semi-implicit two-step projection method is used to solve the NS equation, and the numerical calculation method of the additional term is given. Finally, the calculation process of the numerical model is given. The numerical calculation model combining the immersion boundary method and the fluid volume method is numerically verified, mainly through two examples of linear periodic wave passing through a trapezoidal object and solitary wave passing through a rectangular object. The comparison of the results proves that the numerical model established in this paper is correct and reliable, and can solve the interaction between waves and structures well. At the same time, the numerical model established in this paper is used to calculate the hydrodynamic characteristics of the near-wall cylinder under the action of waves .

Funding

This article is not supported by any foundation.

Data Availability

Data sharing is not applicable to this article as no new data were created or analysed in this study.

Conflict of Interest

The author states that this article has no conflict of interest.

References

[1] Tolstykh A I , Shirobokov D A . Excitation and Development of Instability in a Compressible Boundary Layer as Obtained in High-Order Accurate Numerical Simulation without Introducing Artificial Perturbations. Computational Mathematics and Mathematical Physics, 2022, 62(7):1180-1192. https://doi.org/10.1134/S0965542522070090

[2] FelipePamplonaMariano · LeonardodeQueirozMoreira · AndreiaAoyaguiNascimento · AristeuSilveiraNeto. an improved immersed boundary method by coup1ing of the mu1ti direct forcing and fourier pseudo spectra1 methods. Journal of the Brazilian Society of Mechanical Sciences and Engineering, 2022, 44(9):1-23. https://doi.org/10.1007/s40430-022-03679-5

[3] Sawant V A , Patel J B , Joseph J , et al. An experimental and numerical comparative study on the uplift capacity of single granular pile anchor and rough pile in sand. International Journal of Geotechnical Engineering, 2022, 16(4):499-513. https://doi.org/10.1080/19386362.2021.1999077

[4] Betaieb E , Duchêne, Laurent, Habraken A M . Calibration of kinematic hardening parameters on sheet metal with a Computer Numerical Control machine. International Journal of Material Forming, 2022, 15(5):1-17. https://doi.org/10.1007/s12289-022-01714-3

[5] Shi L , Zhuang M , Zhou Y , et al. Domain decomposition based on the spectral element method for frequency-domain computational elastodynamics. Science China Earth Sciences, 2021, 64(3):388-403. https://doi.org/10.1007/s11430-020-9696-4

[6] Sezen S , Atlar M . An alternative Vorticity based Adaptive Mesh Refinement (V-AMR) technique for tip vortex cavitation modelling of propellers using CFD methods. Ship Technology Research, 2022, 69(1):1-21. https://doi.org/10.1080/09377255.2021.1927590

[7] Zamani L , Sadjadi S , Ashouri F , et al. Carbamazepine removal from aqueous solution by

synthesized reduced graphene oxide-nano zero valent iron (Fe0-rGO) composite: theory, process optimization, and coexisting drugs effects.. Water science and technology : a journal of the International Association on Water Pollution Research, 2021, 84(9):2557-2577. https://doi.org/10.2166/wst.2021.457

[8] Xu Y , Shen G , Zhang Y , et al. Forming numerical analysis and experiment research on hot rolling of straight face gear. The International Journal of Advanced Manufacturing Technology, 2022, 121(11-12):8299-8311. https://doi.org/10.1007/s00170-022-09787-9

[9] Odinokov V I , Evstigneev A I , Dmitriev E A , et al. Mathematical Simulation of CCM Mold Filling with Liquid Metal during Its Feeding from Rotary Submerged Nozzle. Steel in Translation, 2022, 52(2):179-186. https://doi.org/10.3103/S0967091222020164

[10] Meraou M A , Al-Kandari N M , Raqab M Z . Univariate and Bivariate Compound Models Based on Random Sum of Variates with Application to the Insurance Losses Data. Journal of Statistical Theory and Practice, 2022, 16(4):1-30. https://doi.org/10.1007/s42519-022-00282-8

[11] Mcnett M , Tucker S , Zadvinskis I , et al. A Qualitative Force Field Analysis of Facilitators and Barriers to Evidence-Based Practice in Healthcare Using an Implementation Framework. Global Implementation Research and Applications, 2022, 2(3):195-208. https://doi.org/10.1007/s43477-022-00051-6

[12] Plotkin E R , Kontorovich T S . Analysis of the Thermal Stress Concentration Coefficients in Tee Joints. Thermal Engineering, 2022, 69(8):632-635. https://doi.org/10.1134/S0040601522080067

[13] Li M , Cao T , Su C , et al. Geometrical error correction based on sensitivity equations for accurate specular surface measurement with deflectometry. Journal of Modern Optics, 2022, 69(10):554-565. https://doi.org/10.1080/09500340.2022.2067359

[14] Cho W , Park J , Moon J , et al. Effects of topography and sea surface temperature anomalies on heavy rainfall induced by Typhoon Chaba in 2016. Geoscience Letters, 2022, 9(1):1-18. https://doi.org/10.1186/s40562-022-00230-1

[15] Okuda S , Sato K , Hiraiwa T . Continuum modeling of non-conservative fluid membrane for simulating long-term cell dynamics. The European Physical Journal E, 2022, 45(8):1-14. https://doi.org/10.1140/epje/s10189-022-00223-0

[16] Jiang J , Huang Z , Gong R , et al. Time-varying Numerical Simulation Of Reservoir Physical Properties Based On Surface Flux. Journal of Physics: Conference Series, 2021, 1894(1):012077 (6pp). https://doi.org/10.1088/1742-6596/1894/1/012077

[17] Sarkar A , Bora S N . Hydrodynamic force and wave run-up due to diffraction of ocean water waves by a surface-piercing bottom-mounted compound partial-porous cylinder. Fluid Dynamics Research, 2021, 53(1):015508 (19pp). https://doi.org/10.1088/1873-7005/abdb3d

[18] Jeon Y , Park J , Jeon K . A Numerical Study on the Characteristics of the Supercavitation and Hydrodynamic Forces Generated in a Supercavitating Underwater Vehicle with Angle of Attack. Journal of the Society of Naval Architects of Korea, 2021, 58(4):214-224. https://doi.org/10.3744/SNAK.2021.58.4.214