Numerical Computation of Squat of Ships in Waves

Li Qiang, Hong Bi-guang, Gao Xiao-ri

Navigation College, Dalian Maritime University , Dalian 116026,Liaoning, China

Abstract
Numerical wave tank is established on the basis of computational fluid dynamics (CFD) method, and numerical computation and simulation are conducted for the motion state of ships in waves. Comparison validation is carried out between the computation results and experimental data of ships, and they are in in good agreement. A numerical simulation method for ships sailing in waves is deduced; the hydrodynamic force borne by Yu-Kun and the moving posture of the ship in heading waves under different navigational speeds are computed. According to the research, numerical simulation based on numerical wave tank can be realized and controlled more easily when compared with the test. Able to gain the trim angle and squat of ships sailing in heading waves, it has extensive application values for drivers to master the keel clearance.
Key words: SQUAT OF SHIPS, COMPUTATIONAL FLUID MECHANICS, REGULAR WAVE, PITCHING; HEAVING

1. Introduction

In recent years, with the rapid development of computer technology, the application of computational fluid dynamics (CFD) and computer technology have also developed quickly, and it has gained rapid expansion to simulate and compute the motion state of ships by utilizing computational fluid mechanics. With the continuous expansion and application of CFD of ships, numerical wave tank (NWT) [1] technology has also made a progress. Numerical wave tank can carry out motion simulation studies for ships in waves by simulating physical tank.

Compared with physical tank, numerical wave tank has advantages of low cost, non-contact flow field measurement and scale-free effect; besides, it can eliminate the influence of sensor size and model deformation on the flow field in physical model experiment, and acquire detailed flow field information. Therefore, it has drawn much attention and its application range has become wider and wider [2-4]. It has provided an important new approach for hydrodynamic performance studies of ships. In recent years, it has become a focused issue for domestic and overseas researchers to study hydrodynamics of ships via numerical simulation method based on numerical wave tank technology, and an important research progress has been made in the aspects of ship maneuverability, rapidity and ship performance prediction in waves [5-10]. However, different researchers will use different computational methods and realization means. Some of them are not systematic and comprehensive enough due to the simplification of the issue, some researches need to be further improved, and the computation results of some researches should be verified and analyzed. Due to the importance of hydrodynamic performance and motion prediction of ships in waves, targeted, detailed and systematic study is still required. Only in this way, can the squat of ships in waves be accurately predicted.

In this paper, systematic numerical simulation study will be conducted for the motion of ships in regular waves on the basis of numerical wave tank technology. Firstly, a wave environment expression method for numerical simulation of the ship sailing in waves is derived, so as to successfully simulate the regular waves and verify the effectiveness of wave absorbing. Secondly, moving postures of the ship sailing in waves under different navigational speeds are simulated and computed in numerical wave tank, especially the rolling angle; besides, contrastive analysis is made with relevant test data in [11]. Finally, the heaving and pitching motions of the ship model in regular heading waves are simulated and predicted, and squat of the ship in waves of different parameters is worked out.

2. Numerical tank

2.1 Control equation

Based on viscous flow CFD theory and multiphase flow theory, a three-dimensional numerical wave tank similar to physical test tank is established. In a cuboid shape, the upper part of the tank is air and the lower part is water. In this paper, a reference coordinate system fixed on the ship and sailing steadily with the ship is used to simulate and compute the hydrodynamic force borne by the ship in waves. As for the flow field, continuity equation and NS equation are set as control equations.

$$\frac{\partial \rho}{\partial t}+\frac{\partial(\rho u_i)}{\partial x_i}=0 \qquad (1)$$

$$\frac{\partial(\rho u_i)}{\partial t}+\frac{\partial(\rho u_i u_j)}{\partial x_j}=-\frac{\partial P}{\partial x_i}+\frac{\partial}{\partial x_i}\left(\mu\frac{\partial u_i}{\partial x_j}-\rho\overline{u_i' u_j'}\right)+S_i+\rho f_i \qquad (2)$$

Based on viscous flow CFD theory and multiphase flow theory, a three-dimensional numerical wave tank similar to physical test tank is established. In a cuboid shape, the upper part of the tank is air and the lower part is water. In this paper, a reference coordinate system fixed on the ship and sailing steadily with the ship is used to simulate and compute the hydrodynamic force borne by the ship in waves. As for the flow field, continuity equation and NS equation are set as control equations.

In the formula, u_i refers to the velocity component of fluid particle at i direction; f_i means the mass force; S_i indicates the additional momentum source term; P denotes the fluid pressure; ρ signifies the fluid density, defined as $\rho=\sum_{q=1}^{2}a_q\rho_q$ in which the volume fraction; a_q means the ratio between the fluid volume of phase q and the total volume in the unit and $\sum_{q=1}^{2}a_q=1$; μ is the dynamic viscosity coefficient of phase volume fraction average, defined as the same form of density.

The change of free surface in the tank is simulated via VOF (volume of fluid) method, i.e.

$$\frac{\partial a_q}{\partial t}+\frac{\partial(u_i a_q)}{\partial x_i}=0,(q=1,2) \qquad (3)$$

In the formula, a_1 and a_2 refer to the volume fractions of air phase and water phase respectively. When $a_q=0.5$, it is the free surface.

2.2 Numerical wave generation and wave absorbing technology

In order to predict the hydrodynamic force and motion of ships sailing in waves, the wave environment that meets computing requirements should be simulated at first. In another word, numerical wave generation should be realized in the tank. In numerical simulation of this paper, direct input method is adopted. In other words, the fluid flow velocity function is given at the entrance boundary, so as to realize wave generation; this method is simple and feasible. According to linear wave theory, the wave equation and velocity field of regular waves in deep water are as follows:

$$\eta=\xi_0\cos(kx-\omega_0 t) \qquad (4)$$

$$\begin{cases} u=\omega_0\xi_0 e^{kz}\cos(kx-\omega_0 t)\\ v=0\\ w=\omega_0\xi_0 e^{kz}\sin(kx-\omega_0 t)\end{cases} \qquad (5)$$

In the formula, ξ_0 means the wave amplitude; k refers to the wave number; ω_0 denotes the circular frequency of waves. ox, oy and oz are defined as length direction (the direction that the wave advances) of the tank, width direction of the tank and water depth direction respectively. Wave generation in the tank is realized by simulating the flexible wave plate movement at the entrance boundary of the tank, i.e. giving the velocity distribution of fluid particles.

2.3 Expression of waves under reference coordinate system

Suppose that the incident wave faces the ship along the direction of axis. Based on wave theory, the velocity potential of the incident wave in finite depth water under the fixed coordinate system o_0-xyx is:

$$\Phi=A_0\xi_0\frac{\cosh(k(z+H))}{\sinh(kH)}\sin(kx-\omega_0 t) \qquad (6)$$

In the formula, H is the water depth of the tank; A_0 meets the relational expression

$$A_0=\frac{\omega_0}{k}=\sqrt{\frac{g}{k}\tanh(kH)}$$

According to coordinate transformation, when the ship direction of incident wave and axis form the angle of β, the velocity potential can be expressed as:

$$\Phi = A_0\xi_0 \frac{\cosh(k(z+H))}{\sinh(kH)} \sin(k(x\cos\beta + y\sin\beta) - \omega_0 t) \quad (7)$$

The coordinate system $o-x'y'z'$ fixed on the ship and sailing steadily with the ship under U is set as reference coordinate system, and coordinate transformation between reference coordinate system and fixed coordinate system is expressed as:

$$\begin{cases} x = x' + Ut \\ y = y' \\ z = z' \end{cases} \quad (8)$$

Formula (8) is substituted into formula (7), and the velocity potential of incident wave under $o-x'y'z'$ is

$$A_0\xi_0 \frac{\cosh(k(z'+H))}{\sinh(kH)} \sin(k(x'\cos\beta + y'\sin\beta) - \omega_e t) \quad (9)$$

Where the encounter frequency $\omega_e = \omega_0 - kU\cos\beta$

The expression of wave surface in the tank is $\eta = \xi_0 \cos(k(x'\cos\beta + y'\sin\beta) - \omega_e t)$.

Velocity expression of fluid particle under reference system is

$$\begin{cases} u = \omega_0\xi_0 \cos\beta \dfrac{\cosh(k(z'+H))}{\sinh(kH)} \cos[k(x'\cos\beta + y'\sin\beta) - \omega_e t \\ v = \omega_0\xi_0 \sin\beta \dfrac{\cosh(k(z'+H))}{\sinh(kH)} \cos[k(x'\cos\beta + y'\sin\beta) - \omega_e t] \\ r = \omega_0\xi_0 \dfrac{\sinh(k(z'+H))}{\sinh(kH)} \sin[k(x'\cos\beta + y'\sin\beta) - \omega_e t] \end{cases} \quad (10)$$

2.4 Ship motion in waves

When the ship sails in waves, the ship motion and surrounding three-dimensional flow can realize coupling. According to motion theory of ships in waves, the force and moment of force borne by the ship in the tank are:

$$F = \int_S ([\tau] - p[I])ndS - mg \quad (11)$$

$$M_C = \int_S (r - r_C) \times ([\tau] - p[I])ndS \quad (12)$$

In the formula, F and M_C are external force and moment of external force borne by the ship respectively; $[\tau]$ refers to the shear stress component; p means the fluid pressure; n represents the outer normal direction of hull surface; m indicates the ship model mass; r_C denotes the coordinate vector of the ship model mass center and the subscript C means the mass center; r signifies the coordinate vector of any point on the hull surface; g means the acceleration of gravity.

According to Newton's second law, theorem of motion of mass center and theorem of moment of momentum centering on mass center are adopted. Thus the six degrees of freedom kinematics equation of the ship is

$$F = \frac{d}{dt}(mv_C) \quad (13)$$

$$M_C = \frac{d}{dt}([I_C] \cdot \omega_C) \quad (14)$$

In the formula, v_C means the linear velocity vector component of the ship; $[I_C]$ refers to the inertia component; ω_C signifies the angular velocity vector of ship rotation. After the external force and moment of external force acting on the ship are gained, velocity and angular velocity of ship motion can be solved through formula (13) and formula (14).

When the ship moves in the flow field, the hydrodynamic force of fluid on the ship will drive ship motion. In return, the ship motion will affect the surrounding flow field, and these two can realize coupling.

3. Model and grid

3.1 Establishment of computational domain model

On the premise of meeting the test requirements, relative motion coordinate system was adopted in setting of computational domain [5]. The length, width and height of computational domain were set as 3.5L, 2.6L and 1.1L. The distance between the inlet and bow was 1L, the distance between the outlet and bow was 1.5L, the distance between mid-ship section and left and right boundaries was 1.3L, and the distances between free surface and up and down boundaries were 0.1L and 1L, as shown in Figure 1

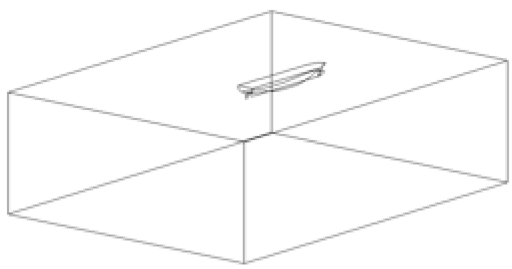


Figure 1. The computational domain model

3.2 Brief introduction to "Yu-Kun"

The teaching-training ship of Dalian Maritime University, "Yu-Kun" was set as numerical simulation test object of this paper. Main parameters of "Yu-Kun" are presented in Table1:

Table 1. The main parameter of "Yu-Kun"

Main Features	Real Vessel	Model
Length overall (m)	116	2.32
Designed waterline length (m)	106.5	2.13
Length between perpendiculars (m)	105	2.1
Ship width (m)	18	0.36
Molded depth(m)	8.35	0.167
Designed draft d(m)	5.4	0.108
Block coefficient Cp	0.56	0.56

3.3 Establishment of hull model and rudder model

When models were established, solid modeling was adopted for both hull and rudder. Hull model and rudder model were established separately.

After the models were established, bulk treatment was conducted according to the actual relative positions of hull and rudder.

Figure 2. The 3D geometry model of hull and rudder by "Yu-Kun"

The structured hexahedral grid was adopted for partitioning of "Yu-Kun" hull grid and rudder grid. Partitioning of hull grid and rudder grid was treated separately.

The structured grids shown in Figure 3 were gained, and the number of hull surface grids is 22,540.

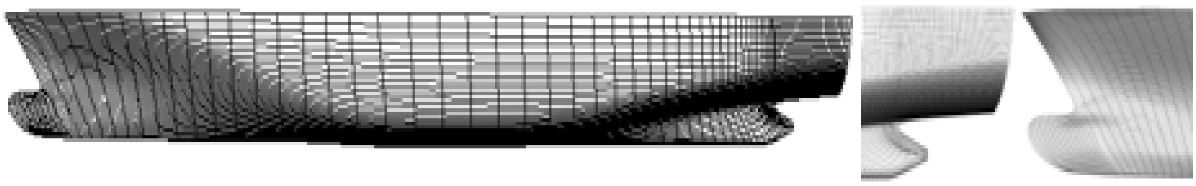


Figure 3. Girds on hull

Overset grid completed transfer of flow field information through interpolation boundary (pore boundary), so body-fitted grid generation treatment should be conducted for grids on hull and rudder surface. In order to increase the overset grid quality, hull and rudder were made to form O-type body-fitted grids through extruding form. According to the Reynolds number of actual ship, dimensionless distance of grid wall at boundary layer was set as y+=60, dimensionless grid thickness at the first layer of hull surface was set as Δyp=2e-6, the growth rate of grids was set as Ratio=1.25, and there were 45 extrapolation layers. As for body-fitted grids of rudder, there were 41 extrapolation layers, and other parameters were the same with hull settings. The number of O-type body-fitted grids on the hull and rudder is 400,050 and 92,742 respectively, as shown in Figure 4.

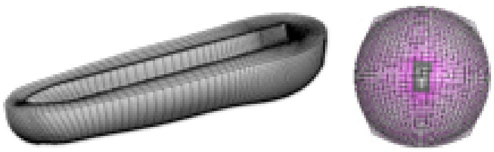


Figure 4. Body-fitted grids

Structured hexahedral grid was adopted for grid partitioning of computational domain. In order to better calculate the viscous flow filed around the ship, grid refinement treatment was conducted on bow, stern and free surface, and the number of grids is 727,056, as shown in Figure 5.

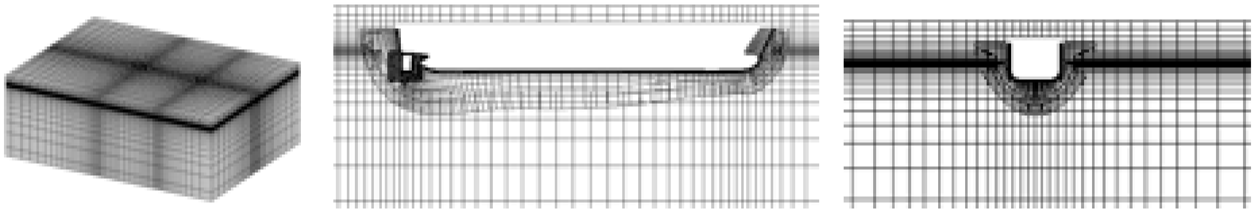


Figure 5. Computational grids and Overset grids

After grid generation was completed for hull, rudder and computational domain, oversetting should be conducted for hull body-fitted grids, rudder body-fitted grids and computational domain grids.

3.4 Setting of boundary conditions

The control equation of fluid motion has infinite possible solutions. For any physical phenomenon, a complete mathematical model should contain control equation and corresponding boundary conditions at the same time. Boundary condition refers to the rule that the solution variable gained at the boundary of solution domain or its first-order derivative changes with position and time. Only by giving reasonable boundary conditions, can the solution of flow field be gained. Therefore, boundary condition is the necessary condition that endows CFD problem with a fixed solution, and all CFD problems have boundary conditions. Boundary conditions of computational domain in this paper are inlet, outlet, wall and open boundaries.

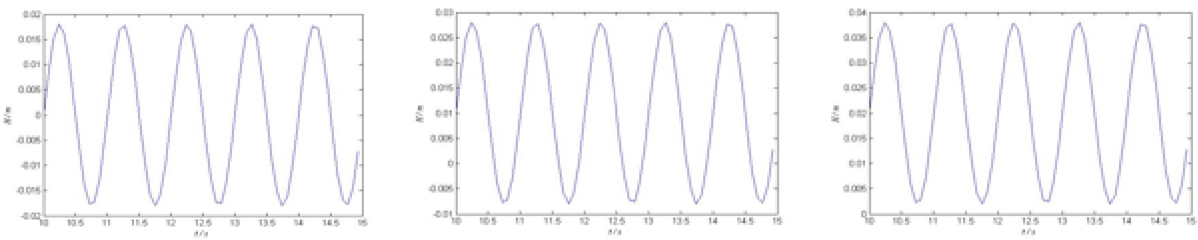
3.5 Boundary conditions and discretization schemes

In numerical simulation, boundary conditions of the tank are set as follows: flexible wave plate movement is simulated at the entrance, and velocity distribution is set according to formula (11); the navigational speed is set at the entrance boundary in the form of fluid velocity; the outlet boundary is set as outflow boundary condition; the top and bottom of the tank are set as velocity boundary condition, and the left side and right side are set as symmetry boundary condition. In order to solve the control equation, discretization should be conducted for the control equation. The discretization method adopted by FLUENT is finite volume method. In this paper, the control equation in the flow field is solved on the basis of FLUENT solver. Moreover, transient term in second-order implicit discrete momentum equation is adopted, and second-order upwind discrete convective term and diffusion term are used. Pressure – velocity coupling is solved with iterative computation by applying SIMPLE method. By compiling user defined function (UDF), numerical wave generation, wave absorbing and velocity boundary setting are realized in FLUENT software.

4. Numerical simulation results and analysis

4.1 Numerical simulation of regular waves

In this paper, simulation and verification are conducted for generation and spreading of regular waves with the cycle of $T = 1.0s$ and wave amplitude of $H = 0.018$, $H = 0.028$ and $H = 0.038$ in numerical wave tank. In the simulation process, monitoring points of wave height are set in the tank, and the time-dependent duration curve of wave surface can be gained .After simulation and computation for a certain period, the simulated regular waves almost reach a steady state and the wave amplitude is identical with the given wave amplitude.



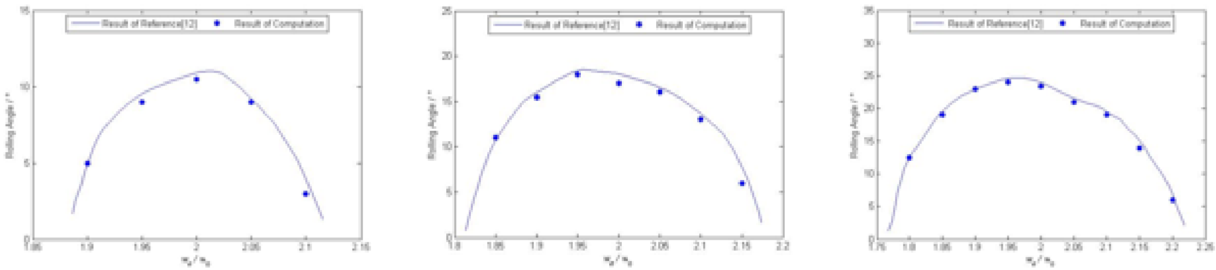
(a) H= 0.018 (b) H= 0.028 (c) H= 0.038

Figure 6. Duration curve of wave surface

4.2 Rolling computation of Yu-Kun

Numerical simulation based on numerical wave tank technology can solve the complicated hydrodynamic force problem of ships, and the prediction result can be included in factors like fluid viscosity and nonlinearity. In order to compute the motion of ship model in waves, the wave force and moment of force borne by the ship should be gained at first. The rolling and pitching angles of the ship in waves are related to factors like intrinsic rolling period of the ship, intrinsic pitching period, wave height, wave length, wave frequency, and wave direction angle. The ratio between wave height and wave length is called wave steepness. The rolling angle of the ship is different under different wave steepness values. Another factor that influences rolling angle of the ship is the ratio between the encounter wave period and intrinsic rolling period of the ship in waves. When the encounter wave period is nearly 2 times more than intrinsic rolling period of the ship, rolling angle of the ship is the largest. In this paper, rolling angles of the ship under different wave steepness values and encounter wave periods are simulated via CFD. Besides, a comparison is made with experimental results in literature [12]. See Figure 7 for the comparison results.

According to Figure 5, when the steepness values are 0.03, 0.05 and 0.07, computation results of this paper fit the experimental data in literature [12] well. Under different steepness values, the computation results are almost identical with the experimental data in literature [12]. In another word, when the encounter wave period is nearly 2 times more than intrinsic rolling period of the ship, rolling angle of the ship is the largest. When the wave steepness is 0.03, and the ratio between the encounter wave period and intrinsic period is smaller than 1.83 and greater than 2.07, the ship cannot reach a stable rolling state and the rolling angle approaches 0 in a certain period. When the wave steepness values are 0.05 and 0.07, similar situations occur during simulation. Therefore, numerical simulation based on CFD method can be expanded, and the prediction results based on CFD



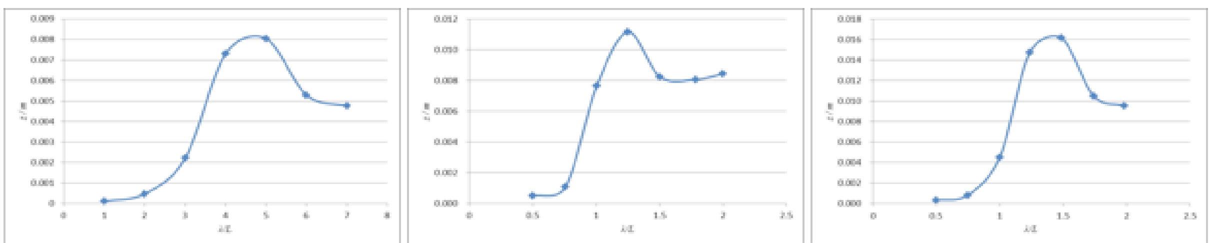
(a) $S_w = 0.03$ (b) $S_w = 0.05$ (c) $S_w = 0.07$

Figure 7. Maximum value of rolling of the ship under different wave steepness values

will be more accurate. Besides, the interaction among incident wave, wave making of ship model sailing, and wave making of radiation can be reflected. Meanwhile, the wave surface is stable and smooth in wave absorbing area of the tank, showing the effectiveness of wave absorbing.

4.3 Motion simulation and computation of ships sailing in heading waves

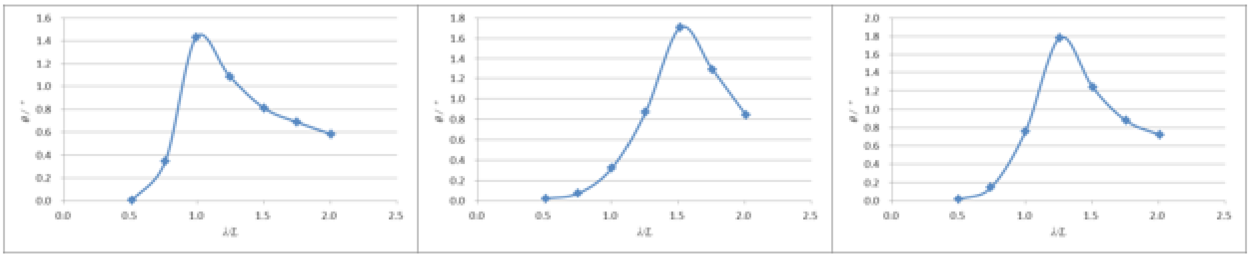
In numerical simulation for the ship motion in regular heading waves, this paper mainly considers heaving and pitching motion modes (the two motions of heaving and pitching are given). Of course, the prediction method of this paper can be easily expanded and applied to the six degrees of freedom motion computation of ships in waves.



(a) $H = 0.018$ (b) $H = 0.028$ (c) $H = 0.038$

Figure 8. Maximum value of rolling of the ship under different wave steepness values

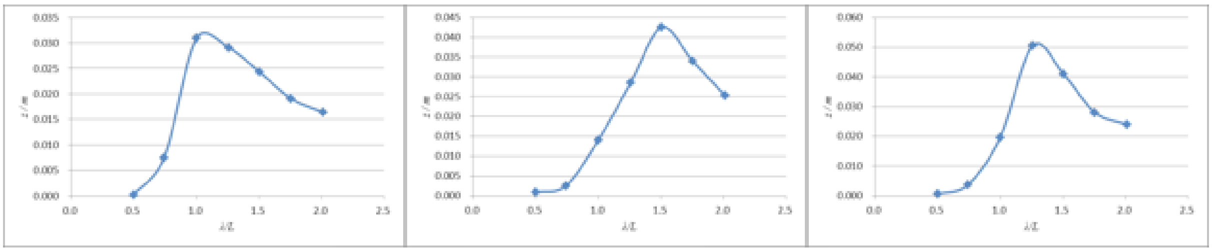
Figure 8 presents the amplitude changing curve of heaving motion of Yu-Kun ship model sailing in regular heading waves under different navigational speeds. According to the figure, under different wave heights, the changing trend of the heaving motion amplitude with the ratio between wave length and ship length is almost consistent. When the ratio between wave length and ship length is 1.2-1.5, heaving amplitude of the ship is the greatest. When the ratio between wave length and ship length is smaller than 0.5, vertical squat of the ship is the smallest.



(*a*) $H = 0.018$ (*b*) $H = 0.028$ (*c*) $H = 0.038$

Figure 9. Pitching amplitude of the ship under different wave heights

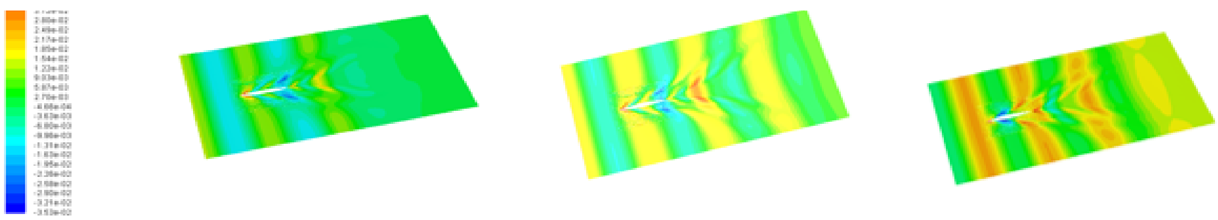
Figure 9 presents the amplitude changing curve of pitching motion of Yu-Kun ship model sailing in regular heading waves under different navigational speeds. According to the figure, under different wave heights, the changing trend of the pitching motion amplitude with the ratio between wave length and ship length is almost consistent. When the ratio between wave length and ship length is 1.2-1.5, pitching amplitude of the ship is the greatest. When the ratio between wave length and ship length is smaller than 0.5, pitching of the ship is the smallest.



(*a*) $H = 0.018$ (*b*) $H = 0.028$ (*c*) $H = 0.038$

Figure 10. Squat of the ship under different wave heights

Squat of the ship in waves is related to heaving and pitching motions of the ship. According to the computation results, heaving motion and pitching motion of the ship happen at the same time. When pitching reaches the maximum value, the heaving amplitude is often the smallest; when heaving reaches the maximum value, the pitching amplitude of the ship approaches 0. The overall squat of the ship is the superposition of heaving value and the increase value of ship draught caused by pitching at the same time.



(*a*) $H = 0.018$ (*b*) $H = 0.028$ (*c*) $H = 0.038$

Figure 11. Free surface waveform with different wave heights

Figure 11 is the cloud picture for free surface waveform of Yu-Kun ship model sailing in regular heading waves under different wave heights. The interaction among incident wave, wave making of ship model sailing, and wave making of radiation can be reflected.

5. Conclusion

Numerical wave tank is established on the basis of CFD method, and systematic numerical computation and simulation are conducted for the motion state of ships sailing in regular waves. Firstly, a wave environment expression method for numerical simulation of the ship sailing in waves is derived, so as to simulate and verify generation, spreading and wave absorbing of regular waves in numerical wave tank. Secondly, the rolling angle of Yu-Kun ship model sailing in regular heading waves under different navigational speeds is simulated and predicted in numerical wave tank. Finally, the heaving and pitching motion modes of the ship sailing in heading waves are predicted. CFD numerical computation results of this paper are compared with corresponding experimental data, and they are in good agreement. According to the systematic research and computation for the hydrodynamic force and motion of the ship sailing in heading waves via numerical wave tank based on CFD method, numerical simulation of this paper can accurately reflect the interaction among incident wave, wave making of ship model sailing, and wave making of radiation, and gain precise prediction results. Thus its applicability and feasibility of predicting ship motion in waves in numerical wave tank are verified. Compared with physical model test, method of this paper is more convenient and efficient; moreover, it has advantages of scale-free effect, low cost, easy measurement and control, and ability to gain detailed flow field information around the ship. Therefore, this method plays an important role in analytical investigation of hydrodynamic performance and motion prediction of ships in waves, and has broad application prospects in hydrodynamic research of ship and ocean engineering structures.

Acknowledgements

This work was supported by “the Fundamental Research Funds for the Central Universities” (01750111).

References

1. Cheng Hongxia, Li Jide (2007) Forced-heaving hydrodynamic computation of any two-dimensional section by FLUENT. *Chinese Journal of Ship Research*, 2(1), p.p.30-33.
2. Gong Jingfeng, Zhang Wenping, Ming Pingjian, et al.(2012) Numerical analysis of the propeller low frequency flow-noise. *Chinese Journal of Ship Research*, 7(5), p.p.14-21.
3. Fang Zhaozhao, Zhu Renchuan, Miao Guoping (2011) Numerical simulation on radiation problems of moving vessels in numerical wave tank. *Chinese Journal of Hydrodynamics*, Ser. A, 26(1), p.p.11-19.
4. Kaeding Y X, Jensen G, Hadzie I, et al. (2004) Simulation of flow induced ship motions in waves using a RANSE method. *Journal of Ship Technology Research*, 51(2), p.p.56-68.
5. Luquet R, Gentaz L, Ferrant P, et al. (2004) Simulation of the viscous flow past a ship in waves using the SWENSE approach. *Proceedings of the 24th Symposium on Naval Hydrodynamics*, St Johns, Newfoundland.
6. Wu Chengsheng, Zhu Dexiang, Gu Min (2008) Computation of hydrodynamic forces for a ship in regular heading waves by a viscous numerical wave tank. *Journal of Ship Mechanics*, 12(2), p.p.168-179.
7. Fang Zhaozhao, Zhu Renchuan, Miao Guoping, et al. (2012) Numerical calculation of hydrodynamic forces for a ship in regular waves based on numerical wave tank. *Chinese Journal of Hydrodynamics*, Ser. A, 27(5), p.p.515-524.
8. Guo Haiqiang, Zhu Renchuan, Miao Guoping, et al.(2008) Estimation of hydrodynamic coefficients of ship in numerical wave tank. Shipbuilding of China, 49(S1), p.p.58-65.
9. Fang Zhaozhao, Zhu Renchuan, Miao Guoping, et al.(2012) Numerical simulation of diffraction problems of moving vessels in numerical wave tank. *Journal of Shanghai Jiaotong University*, 46(8), p.p.1203-1209.
10. Journée J M J.(1992) Experiments and calculations on 4 Wigley hull forms in head waves. Netherlands:Report of Delft University of Technology, Ship Hydrodynamics Laboratory..
11. Qi Peng, Wang Yongxue (2003) 3D numerical-wave-tank technology and its application.

Journal of Dalian University of Technology, 43(6), p.p.825-830.
12. Su Zuojing, Zhang Xianku (2012) The numerical simulation and analysis of parametric rolling for vessel “YU KUN“. *Journal of Harbin Engineering University*, 33(5), p.p.590-594.

Measurement of Industrial Alcohol Concentration Capacitance Method and AIA-GA Nonlinear Correction

Wang Zemin, Li Junxiang

School of Electronic Information Engineering, Xi'an Technological University, Xi'an, 710021, China

Abstract
For the nonlinear output of capacitance sensor in the process of measuring industrial alcohol concentration and the lack of traditional genetic algorithms for nonlinear fitting, Propose an improved immune genetic algorithm. This approach combines the advantages of the artificial immune algorithm(AIA) and a genetic algorithm(GA) in the global search ability and convergence efficiency, introduce improved artificial immune mechanisms in genetic algorithms, to overcome its inherent precocity, low efficiency inadequate. Experimental results show that, compared with traditional methods, this method can effectively eliminate the influence of non-target parameter to sensor output, deal with the nonlinear problems efficiently .
Key words: CAPACITANCE METHOD, INDUSTRIAL ALCOHOL CONCENTRATION, GENETIC ALGORITHM, ARTIFICIAL IMMUNE ALGORITHM.

1. Introduction

Industrial alcohol is the basis of organic chemical raw materials and high-quality fuel, widely used in printing, electronics, hardware, spices, chemical synthesis, etc. Different areas of industrial alcohol concentration requirements vary, and accuracy requirements are high, detection of industrial alcohol concentration in general use of alcohol meter testing, the principle is in accordance with the solution density is detected, the method error is large, the accuracy is relatively limited. Because of the advantages of simple structure, high accuracy, fast response, and so on, the capacitance method has been widely used in the on-line detection of liquid [1].

As same as other sensors, capacitance method is also a serious problem with the nonlinear output[2]. Genetic Algorithm (abbreviated GA) as a traditional nonlinear global optimization method, the main feature is not dependent gradient information, especially for dealing with complex issues and problems in nonlinear. But it is easy to premature convergence to local optima, and the phenomenon of "premature"[3]. Artificial immune algorithm (AIA) is the function of the reference biological immune system, AIA based on the concentration of antibody breeding strategy can effectively maintain the diversity of the population. In this paper, we introduce the relevant steps of AIA into GA algorithm, and make corre sponding