Scholar Publishing Group
Frontiers in Ocean Engineering
https://doi.org/10.38007/FOE.2021.020203
ISSN 2790-2706 Vol. 2, Issue 2: 17-25

Numerical Pool of Marine Engineering Ship Based on CFD

Bulychev Nikolay*

Newcastle Univ, Framlington Pl, Newcastle Upon Tyne NE2 4HH, Tyne & Wear, England

**corresponding author*

Keywords: Computational Fluid Dynamics, Ocean Engineering, Numerical Pool, Numerical Simulation

Abstract: With the development of computers, there are more and more studies on marine engineering and ship numerical pools using computational fluid dynamics (CFD), and the accuracy of numerical simulation prediction is also continuously improved, which provides a lot of convenience for the initial design and verification of ships, and also saves money. A lot of money and model test time, therefore, this paper studies marine engineering and marine numerical pool based on CFD. In this paper, the numerical pool and CFD workflow are briefly described first, then the model is constructed from the control equation, VOF model and stability calculation, and finally the model is analyzed from the parameters of rolling and model resistance.

1. Introduction

With the development of the times, marine resources have become the focus of attention of various countries [1]. The numerical pool simulation platform can be used for the related simulation of wind and wave current experiments in marine engineering, and can also analyze the specific causes of fluid flow phenomena by quantifying the dynamic situation of the tiny flow field of the ship [2]. The numerical pool simulation platform is built by using the world's leading high-performance supercomputer. In terms of flow field observation, it has much finer observation capabilities than physical pools, and the observation direction is relatively free, which is convenient for the actual situation of ships combined with marine wind and wave weather conditions. to select the appropriate best route simulation [3-4]. The use of numerical calculation methods is of great scientific significance in the comprehensive assessment of ship performance [5].

In recent years, many scholars have conducted in-depth discussions and research on marine engineering and ship numerical pools, and have achieved good research results. For example, Anyfantis K N used multiple nonlinear regression (MNLR) and CFD to predict the discharge coefficients of various types of weirs. The results of experimental studies and numerical codes were used to develop the hydraulic characteristics of top weirs with different head heads. Whether it is a rectangular slot weir or a triangular slot weir, the increase of the peak head will lead to an increase of the discharge coefficient. CFD can simulate the hydraulic characteristics of the open channel flow upstream of the weir in the rectangular and triangular gaps [6]. Bayramolu K and other researchers used CFD technology to study the influence of fairing on the hydrodynamics of marine risers, and analyzed the frequency of change of lift coefficient and drag coefficient. The research shows that the riser with fairing has good performance and lift The rms value of the coefficients increases with the chord thickness ratio [7]. At present, although many researchers have conducted research on marine engineering and marine numerical pools, there are relatively few studies on marine engineering and marine numerical pools based on CFD. The use of CFD is beneficial to the research of numerical pools.

This paper studies the numerical pool of marine engineering ships based on CFD, so the structure of this paper is roughly divided into three parts: the first part is a basic elaboration of related theories, such as the introduction of CFD and numerical pool theory; the second part is model construction In this part, the control equation, VFO model and stability calculation are mainly constructed and analyzed; the third part is the analysis of the model, and the model analysis part briefly analyzes the parameter roll and model resistance through CFD numerical simulation analyze.

2. Related Theories

2.1. Numerical Pool

Numerical pool is a high-performance computing cluster as the carrier, CFD as a tool, using self-developed software and secondary development on the basis of open source commercial software. The integrated software system of numerical pool is developed [8]. The numerical pool is to encapsulate the knowledge of various subdivided attributes, and conduct experiments on a class of problems in a simulated way [9]. During the experiment, all parameter settings should be fixed without too many changes, and the process should be adjusted automatically to avoid human intervention as much as possible [10]. The numerical pool is an integrated service platform. The integrated software is the numerical simulation experiment software for ships in the numerical pool. Different simulation software systems can be used individually or as a whole. For the research on different aspects of ships, independent sub-systems can be used. Carry out experiments, and conduct overall experiments and analysis through the integration of sub-systems for the overall study of the ship. The composition of the numerical pool function is shown in Figure 1.

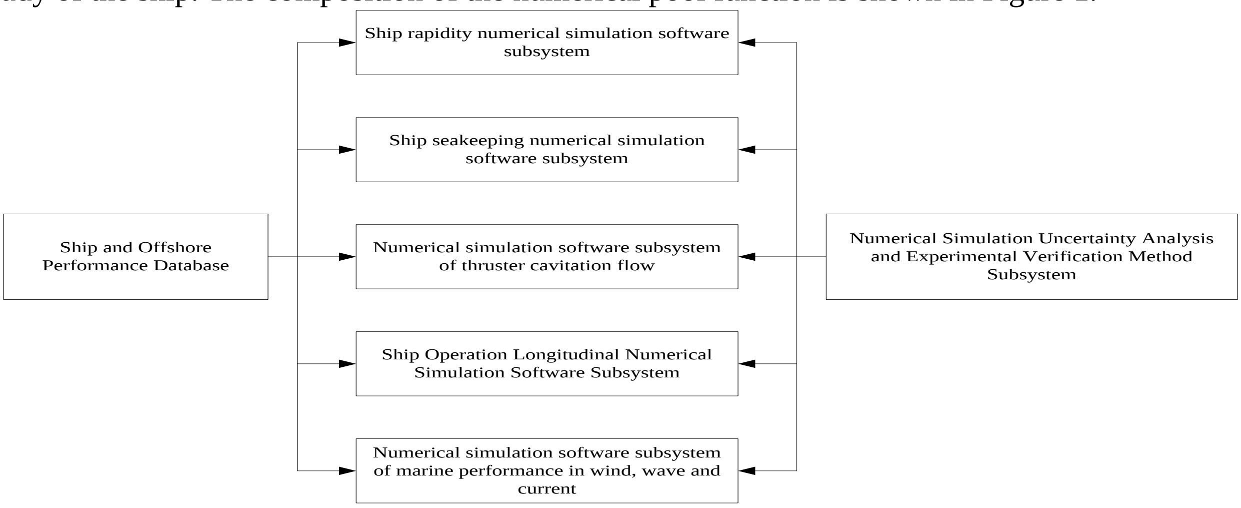

Figure 1. Functional structure diagram of the numerical pool

2.2. CFD Workflow

CFD is highly adaptable and widely used. First, the solution equations for fluid mechanics problems are more independent and nonlinear. The simulation model involves complex computations and numerous boundary parameters, and it is difficult to solve these problems [11]. However, the calculation results that meet the engineering requirements can be obtained by the CFD method. Second, the individual values can be simulated by a computer. For example, the validity and sensitivity of the fluid equations are tested by different parameters in order to compare the various methods [12]. CFD usually studies specific problems such as abstracting objects, establishing models, and setting solving parameters. Similar to single-piece testing, it focuses on new problem solving methods [13-14]. The basic flow of CFD solution is shown in Figure 2.

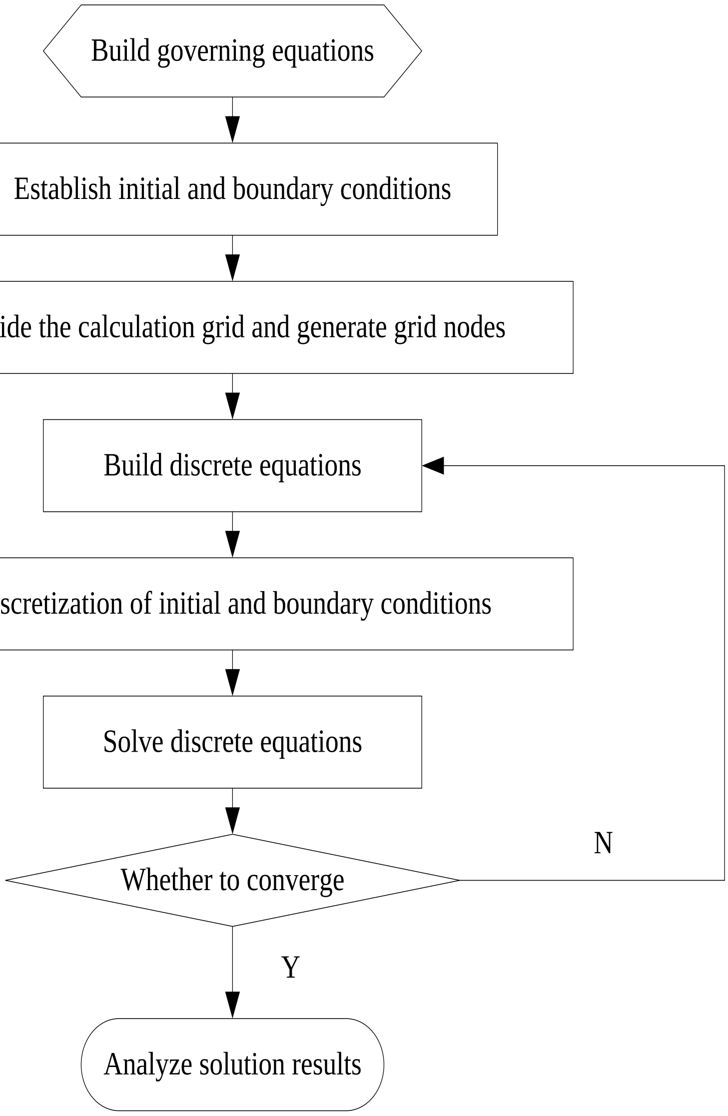

Figure 2. CFD workflow

As shown in Figure 2, the following is a detailed description of the CFD solution steps. The first step is to establish the control equation, which is to mathematically describe the problem under study. After establishing the initial conditions and boundary conditions, the second step begins. The second step is to divide the overall model grid, and the granularity of the calculation is determined by dividing the grid. The third step is to discretize the initial conditions and boundary conditions and solve the discrete equation after establishing the discrete equation. The fourth step is to judge whether the equation is converged. If the equation converges, analyze the solution result. If the equation does not achieve the convergence effect, proceed to the second step until the equation converges.

3. Model Building

3.1. Governing Equations

According to the principle of the law of conservation of mass, the fluid continuity equation is:

$$\frac{\partial \rho}{\partial t}+\frac{\partial \rho e}{\partial x}+\frac{\partial \rho f}{\partial y}+\frac{\partial \rho g}{\partial z}=0 \tag{1}$$

In the formula, e, f, and g represent the velocity components of the fluid on the x, y, and z axes, respectively, and ρ represents the density.

The fluid studied in this paper is a homogeneous incompressible fluid. Formula (1) is simplified in the Cartesian coordinate system to obtain formula (2). Formula (2) is as follows:

$$\frac{\partial e}{\partial x}+\frac{\partial f}{\partial y}+\frac{\partial g}{\partial z}=0 \tag{2}$$

3.2. VOF Model

The numerical pool studied in this paper has an interface between water and air, which involves a two-phase medium of water and air, which belongs to a two-phase flow problem. The key to the numerical simulation study of the flow with free surface lies in the tracking of the free surface [18]. When dealing with two-phase flow problems in ships, the VOF model has unique advantages over other models in dealing with free surface tracking simulations. In this paper, the VOF (Volume of Fluid) multiphase flow model is used to track the free surface, and the gas and liquid two-phase flow calculation is used in this paper. In each calculation unit, the sum of the volume fractions of water and air is 1, which can be expressed as:

$$R_d + R_m = 1 \tag{3}$$

In formula (3), Rd and Rm are the volume fractions of water and air, respectively. In each computing cell, there are three cases: Rd=1 means the cell is completely filled with water; Rd=0 means the cell is completely filled with air; 0<Rd<1 means the cell contains a water-air interface.

3.3. Verification of Stability Calculations

In order to test the reliability of the turbulence model and the numerical pool forced heeling experiment for the stability calculation of the ship when sailing with the waves in regular waves, the ship is simulated, and the same wave conditions as the pool experiment are set, and the wave parameter is a wavelength of 2 meters. , the wave steepness is 0.06, and the Deruder number Fr is 0.1. The comparison between the numerical calculation and the experimental results is shown in Figure 3.

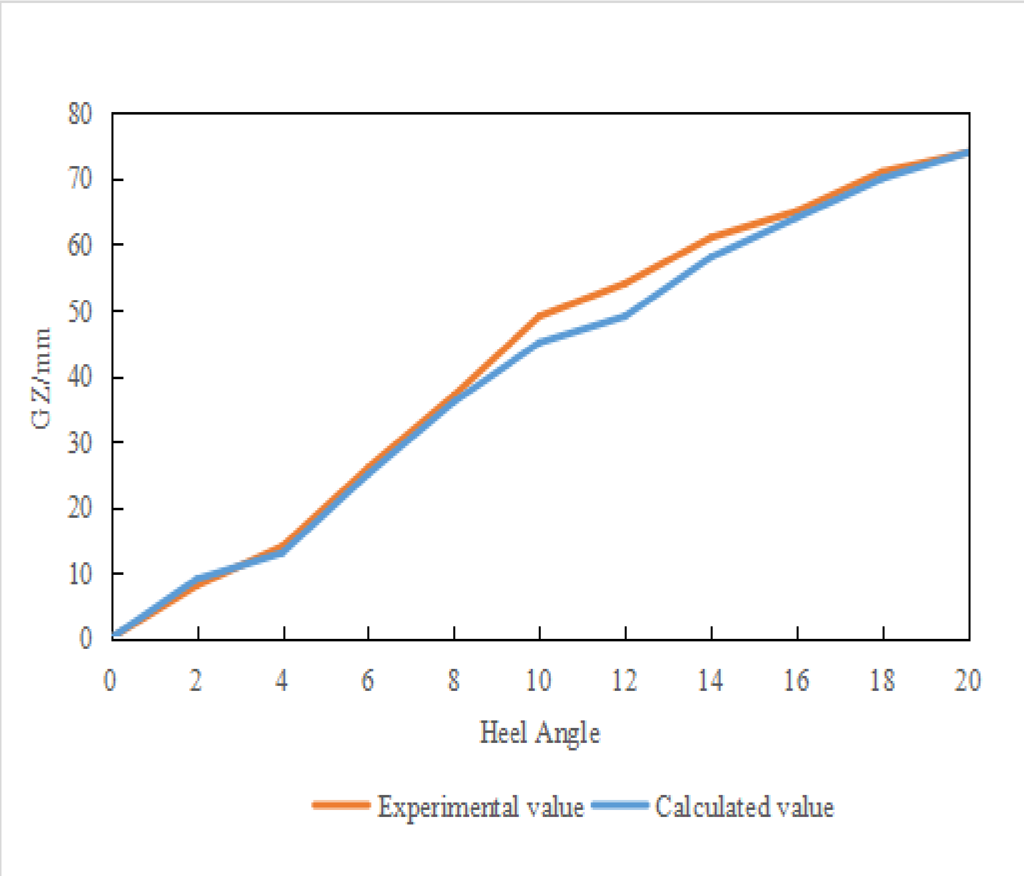

Figure 3. Comparison of wave stability numerical results

Through the analysis of the data in Figure 3, it is found that the numerical simulation results have the same general trend as the results obtained from the pool experiment. When the heel angle is between 0 degrees and 8 degrees and between 16 degrees and 20 degrees, the error between the calculated simulation value and the experimental value is not much different, and the curve changes basically coincide; but when the heel angle is between 8 degrees and 16 degrees, The experimental values are always larger than the numerical simulation results, the reason may be that the conditions in the flow field are idealized when using CFD for numerical simulation. It can be seen from Figure 3 that the maximum difference occurs when the heel angle is 12 degrees, the error between the calculated value and the experimental value is about 9.26%, and the maximum error rate is below 10%. The wave stability analysis of ships while sailing is reliable.

4. Model Analysis

4.1. Parameter Roll Influence Analysis

The parametric roll of the ship often needs to go through multiple rolling cycles under the set working conditions. The excessively long development time of the parametric roll is very unfavorable for the CFD numerical simulation with a large number of grids, a small time step and a large number of iterations. In order to speed up the process of the parametric roll reaching the stable stage and reduce the calculation cost, the roll angular velocity is selected as the initial disturbance, and then the rolling motion of the ship in the headwater of the numerical pool is simulated to the stable development stage of the parametric roll. Period, maximum amplitude and stable roll amplitude to analyze the effect of initial disturbance on parametric roll. For the head wave with a wave steepness of 0.06, the working condition of the ship's Deruder number Fr=0.1 is selected. When the ship model is released, the initial roll angular velocities of 0.07rad/s and 0.7rad/s are applied to the ship model respectively. Statistics are shown in Table 1.

Table 1. Parameter roll data statistics table

Initial Angular Velocity(rad/s)	Cycles Required To Reach Stability	Stable Roll Amplitude	Maximum Roll Amplitude
0.07	23	25.31	29.64
0.7	10	25.12	28.31

It can be seen from Table 1 that the initial roll angular velocity of 0.07rad/s is applied to the ship after 23 cycles, and a stable roll amplitude value of 25.31 is reached. After the initial roll angular velocity of 0.7rad/s, only after 10 roll cycles, the inward-inclined ship reached a stable roll stage, and the roll amplitude was 25.12, of which the maximum roll amplitude was 28.31. By comparison, it is found that the initial roll angular velocity of 0.07rad/s and 0.7rad/s does not affect the roll amplitude after the parametric roll is linearly stabilized, but only shortens the time required for the parametric roll to stabilize, which greatly reduces CFD. The time consumed by the numerical simulation shows that the parametric roll phenomenon can be accelerated by applying a large initial roll angular velocity to the ship, which greatly reduces the time required for the CFD simulation.

4.2. Model Resistance Analysis

CFD simulation was used to calculate the static water resistance at five different speeds when the ship was fully loaded and the draft. The Froude numbers Fr corresponding to the five speeds were 0.108, 0.117, 0.126, 0.135, and 0.144, respectively. After the model is imported, it is checked and a series of settings are performed. After the setting is completed, the calculation will start. The calculation takes more than 50 hours. After the resistance curve is stabilized, the average resistance value for a period of time is calculated as the calculation result. For the computer memory problem, only the most recent 10 results are kept as statistics, and the average value is calculated as the calculation result of this time. The resistance results obtained by CFD numerical simulation are shown in Table 2. In order to observe the error between the calculated resistance value and the experimental value more intuitively, the calculated results are compared with the experimental values, and the comparison results are shown in Figure 3.

Table 2. Resistance calculation result table

Fr	Calculated value(KN)	Experimental value(KN)	Error(%)
0.108	18.32	18.78	-2.44
0.117	23.46	24.23	-3.18
0.126	27.86	27.82	0.14
0.135	34.75	34.13	1.82
0.144	39.14	37.69	3.85

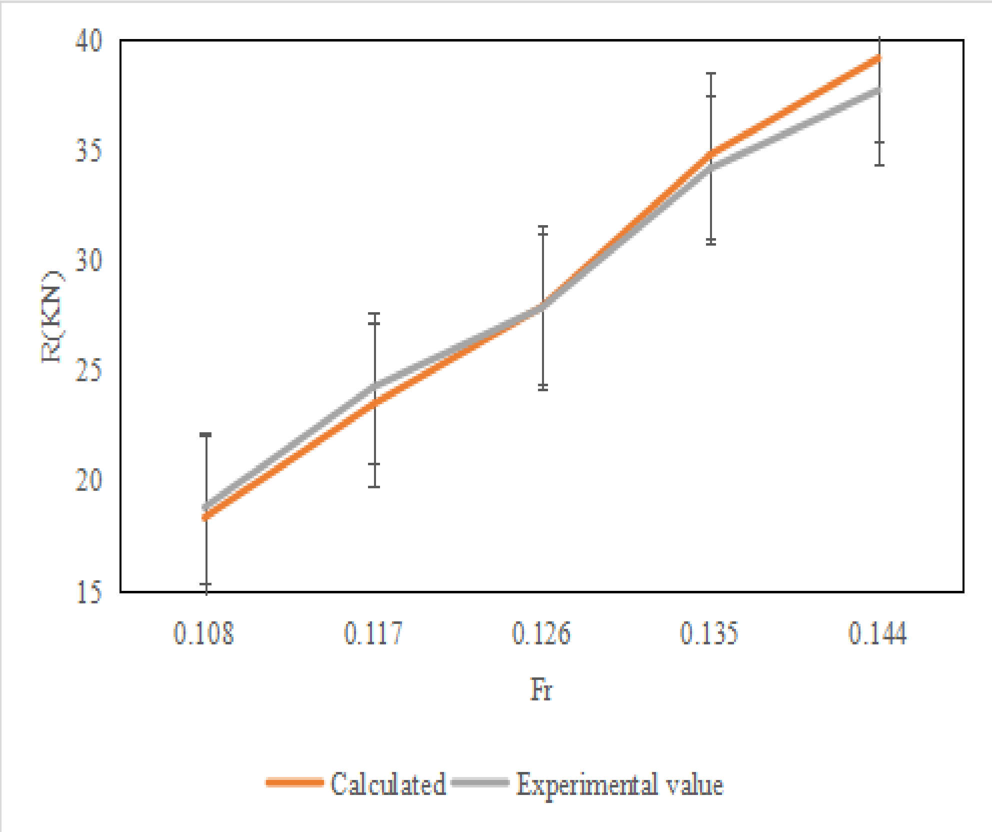

Figure 4. Resistance comparison chart

It can be seen from Table 2 and Figure 4 that when the Fr are 0.108, 0.117, 0.126, 0.135,0.144, the absolute values of the errors between the calculated and experimental values are 2.44%, 3.18%, 0.14%, and 1.82%, respectively. 3.85%. When Fr is 0.126, the numerical simulation error is the smallest, and when Fr is 0.144, the numerical simulation error is the largest, but the error degree of CFD numerical simulation is between 0.1% and 4%. Therefore, CFD numerical simulation can meet the requirements of engineering accuracy. The built model, mesh division, boundary and simulation settings are reasonable.

5. Conclusion

CFD is used more and more widely in the field of marine engineering and ship research. Therefore, this paper studies the numerical pool of marine engineering ships based on CFD, and analyzes the influence of parameter roll. It is found that by applying a large initial roll angular velocity to the ship, the parameters can be accelerated. The roll phenomenon occurs and reduces the time required for CFD simulations. The resistance analysis of the model shows that the CFD numerical simulation can meet the requirements of engineering accuracy and the division of the mesh and the settings of the boundary and simulation are reasonable. The research on numerical pools in this paper is relatively simple and not deep enough, and there are many things that need to be improved. However, the research on numerical pools of marine engineering ships based on CFD is a worthy research direction.

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] Gupta P , Haleem A , Javaid M . Designing of a Carburettor Body for Ethanol Blended Fuel by using CFD Analysis tool and 3D Scanning Technology. Journal of Scientific & Industrial Research, 2019, 78(7):466-472.

[2] Uffelen L , Miller J H , Potty G R . Underwater acoustics and ocean engineering at the University of Rhode Island. The Journal of the Acoustical Society of America, 2019, 145(3):1707-1707. https://doi.org/10.1121/1.5101260

[3] Feng A , Magee A , Price W G . Experimental and Numerical Study for Drillship Moonpool Gap Resonances in Stationary and Transit Conditions in Wave Flume. Journal of Offshore Mechanics and Arctic Engineering, 2019, 142(2):1-26. https://doi.org/10.1115/1.4045623

[4] Shimada K , Arahira T , Matsuno S . ItemSB: Itemsets with Statistically Distinctive Backgrounds Discovered by Evolutionary Method. International Journal of Semantic Computing, 2022, 16(03):357-378. https://doi.org/10.1142/S1793351X22420028

[5] Ovchinnikov K D . Numerical simulation of motions of ship with moonpool in head waves. Proceedings of the Institute for System Programming of RAS, 2018, 30(5):235-248. https://doi.org/10.15514/ISPRAS-2018-30(5)-14

[6] Anyfantis K N . An abstract approach toward the structural digital twin of ship hulls: A numerical study applied to a box girder geometry:. Proceedings of the Institution of Mechanical Engineers, Part M: Journal of Engineering for the Maritime Environment, 2021, 235(3):718-736. https://doi.org/10.1177/1475090221989188

[7] Bayramolu K , Yilmaz S , Kaya D . Numerical and Theoretical Thermal Analysis of Ship Provision Refrigeration System. Journal of ETA Maritime Science, 2019, 7(2):137-149. https://doi.org/10.5505/jems.2019.30922

[8] Seo M G , Bo W N Kim Y . Numerical Evaluation of Ship Turning Performance in Regular and Irregular Waves. Journal of Offshore Mechanics and Arctic Engineering, 2019, 142(2):1-31. https://doi.org/10.1115/1.4045095

[9] Sharma S , Ramakrishna S A , Ramkumar J . Numerical Simulation of Melt Hydrodynamics in Laser Micro Processing of Metals. Procedia CIRP, 2020, 95(1):944-949. https://doi.org/10.1016/j.procir.2020.01.186

[10] Shaikhli H I A , Kadhim K N . Development an equations for flow over weirs using MNLR And CFD simulation approaches. International Journal of Civil Engineering and Technology, 2018, 9(3):70-79.

[11] Handani D W , Ariana I M , Nugroho T F , et al. AIS Based Spatial Distribution of Ship Emission in Madura Strait Indonesia. Journal of the Japan Institute of Marine Engineering, 2018, 53(3):386-391. https://doi.org/10.5988/jime.53.386

[12] MD Arifin, Felayati F M . Numerical Study of B-Screw Ship Propeller Performance: Effect of Tubercle Leading Edge. International Journal of Marine Engineering Innovation and Research, 2021, 6(1):16-23. https://doi.org/10.12962/j25481479.v6i1.8702

[13] Fatsis A . Gas turbine performance enhancement for naval ship propulsion using wave rotors.

Journal of Marine Engineering & Technology, 2021(4):1-13.

[14] Kim B S , Hwang H G , Yoon S W , et al. Development of Transporter for Marine Leisure Ship with Safety and Operation Support System. Journal of Ocean Engineering and Technology, 2019, 33(5):486-494. https://doi.org/10.26748/KSOE.2019.071

[15] Ismail R , Tauviqirrahman M , Mulyana D , et al. Redesigning of 4 (Four) Blades Propeller Installed in a Wooden Fishing Boat in a Ship Yard in Tegal, Central Java Province. Marine Fisheries Journal of Marine Fisheries Technology and Management, 2019, 10(2):187-192. https://doi.org/10.29244/jmf.v10i2.30845

[16] Persaud R , Li H J , Leng J X . Numerical simulation of a passive heave compensator for scientific research ships. Journal of Naval Architecture and Marine Engineering, 2019, 16(1):45-59. https://doi.org/10.3329/jname.v16i1.39960

[17] Geneidy R E , Otto K , Ahtila P , et al. Increasing energy efficiency in passenger ships by novel energy conservation measures. Proceedings of the Institute of Marine Engineering, Science and Technology, 2018, 17(2):85-98. https://doi.org/10.1080/20464177.2017.1317430

[18] Prabowo A R , Baek S J , Byeon J H , et al. Finite element analysis for estimating steel structure responses under a variety of marine-collision actions. International Journal of Earthquake and Impact Engineering, 2018, 2(3):248-267. https://doi.org/10.1504/IJEIE.2018.093425