Computational fluid dynamics-based ship energy-saving technologies
A comprehensive review

Wang, Kai; Li, Zhongwei; Zhang, Rui; Ma, Ranqi; Huang, Lianzhong; Wang, Zhuang; Jiang, Xiaoli

DOI
10.1016/j.rser.2024.114896

Publication date
2025

Document Version
Final published version

Published in
Renewable and Sustainable Energy Reviews

Renewable and Sustainable Energy Reviews 207 (2025) 114896

Contents lists available at ScienceDirect

Renewable and Sustainable Energy Reviews

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

Computational fluid dynamics-based ship energy-saving technologies: A comprehensive review

Kai Wang [a,*,1], Zhongwei Li [a,1], Rui Zhang [a], Ranqi Ma [a], Lianzhong Huang [a,**], Zhuang Wang [c,d], Xiaoli Jiang [b]

[a] Marine Engineering College, Dalian Maritime University, Dalian, 116026, China
[b] Faculty of Mechanical, Maritime and Materials Engineering, Delft University of Technology, Mekelweg 2, Delft, the Netherlands, 2628 CD
[c] Department of Mechanical Engineering, National University of Singapore, Singapore, 117575
[d] Collaborative Innovation Center for Advanced Ship and Deep-Sea Exploration, Shanghai Jiao Tong University, Shanghai, 200240, China

A R T I C L E I N F O

Keywords:
Ship energy efficiency
CFD approaches
Hull optimization design
Drag reduction technology
Navigation state optimization
Energy-saving equipment
Low-carbon shipping

A B S T R A C T

The research on the dynamics analysis-based energy-saving technology is significant to reduce ship energy consumption and greenhouse gas emissions. The adoption of dynamics analysis theory and Computational Fluid Dynamics (CFD) approaches can achieve the optimal design and energy efficiency improvement of the ship. This research focuses on the ship energy efficiency improvement technology through CFD-based dynamics analysis, including the hull optimization design, drag reduction technology, navigation state optimization, efficient propulsion devices, energy-saving equipment, and the coupled dynamics analysis for comprehensive performance optimization. The current research and application status of ship performance optimization based on CFD approaches for energy-efficient shipping are systematically analyzed. On this basis, the challenges and problems in the application of the CFD-based energy-saving technology are discussed, and the future research works are proposed, aiming to provide references for the development of ship energy-saving technology based on CFD approaches. The analysis results show that the adoption of CFD-based dynamics analysis methods can effectively optimize the ship dynamics performance, thus reducing ship energy consumption and pollution gas emissions. In the future, the CFD-based coupled dynamics analysis should be further studied to achieve the overall performance optimization of the integrated ship-engine-propeller-appendages system under the influence of multiple complex factors, to continuously improve the ship energy efficiency, thus promoting the low-carbon development of the shipping industry.

Abbreviation:

CFD	Computational Fluid Dynamics
GHG	Greenhouse gas
IME	Improved maximum entropy
MIGA	Multi-island genetic algorithm
SBD	Simulation-based design
CAD	Computer Aided Design
RSM	Response Surface Model
MIS	Marine creature-Inspired Surface
WAIP	Winged Air Induction Pipe
ALDR	Air-Layer Drag Reduction
VOF	Volume of Fluid
BEM	Boundary element method
TPTR	Twin-propeller twin-rudder
VPP	Variable pitch propellers
AUV	Autonomous Underwater Vehicle
PSS	Pre-swirl stators
GRS	Gate Rudder System
CPRS	Composite propeller-rudder system
DBN	Deep belief network
SPP	Surface-piercing propeller
NSGA II	Non-dominated Sorting Genetic Algorithm II
IMO	International Maritime Organization
ML	Machine learning

(continued on next page)

* Corresponding author.
** Corresponding author.
E-mail addresses: kwang@dlmu.edu.cn (K. Wang), huanglz@dlmu.edu.cn (L. Huang).
[1] These authors contributed equally to this work.

https://doi.org/10.1016/j.rser.2024.114896
Received 2 February 2024; Received in revised form 11 August 2024; Accepted 2 September 2024
Available online 18 September 2024

(*continued*)

SBO	Sampling-based Optimization
FFD	Free-form deformation
PSO	Particle Swarm Optimization
DOE	Design of Experiment
DPD	Dissipative particle dynamics
DRC	Drag reduction coatings
MBDR	Microbubble Drag Reduction
GILLS	Gas-injected liquid lubrication system
ALS	Air lubrication system
DP	Dynamic programming
CRP	Counter-rotating propellers
FPP	Fixed-Pitch Propellers
RD CRT	Rim-driven Counter-Rotating Thrusters
FEM	Finite Element Method
PBCF	Propeller boss-cap-fin
PRS	Propeller-rudder system
AIAD	Artificial intelligence-aided design
ANN	Artificial neural networks
DFOC	Daily fuel oil consumption

1. Introduction

With the continuous advancement of the shipping industry, carbon emissions from maritime transport are also increasing significantly, which is estimated to increase by 250 % by 2050 [1]. Therefore, it is urgent to seek effective ways to decrease carbon emissions [2,3]. As a consequence, the energy-saving and emission-reduction technologies have appealed to widespread concentration [4]. The International Maritime Organization (IMO) has introduced various policies related to maritime emissions to decrease the emission of the greenhouse gas (GHG) from ships [5,6]. Faced with the increasingly serious issues of energy shortages and environmental pollution, the energy saving and emissions reduction have become the primary challenge confronting the development of the ship engineering, and it has a significant effect on promoting the advancement of ship energy-saving technologies [7,8].

Ship energy efficiency enhancement technologies can not only reduce ship energy consumption and carbon emissions effectively [9,10], but also have significant implications for reducing shipping costs and promoting the sustainable advancement of the shipping business [11,12]. However, further studies and applications of innovative technologies for ship energy efficiency improvement are needed to meet the increasingly serious GHG emission requirements [13]. Among them, energy saving and consumption reduction through CFD-based dynamics analysis is an important research direction for ship energy efficiency improvement [14]. The studies and applications of the CFD approaches on ships can provide a highly effective method for enhancing ship energy efficiency [15]. By using CFD approaches, ship hydrodynamic models can be established to solve fluid dynamics control equations, obtain a discrete quantitative description of the flow field, and calculate and predict ship hydrodynamics performance, which can allow for ship optimization design and performance enhancement [16]. In addition, the adoption of CFD approaches can accurately estimate ship dynamics performance parameters with advantages of high accuracy, low cost, and high efficiency [17,18]. Currently, CFD approaches are widely applied in ship flow field analysis and resistance calculation [19,20], ship hull optimization design [21,22], efficient propulsion device design and performance analysis [23], appendage resistance calculation [24], energy-saving equipment and drag reduction [25,26], as well as ship maneuvering optimization [27,28], and have achieved good application results [29].

Although the CFD analysis approaches have been studied and applied in various aspects for ship fuel efficiency optimization, there is still a shortage of a comprehensive analysis for the ship energy efficiency enhancement stemming from dynamics analysis methods, making it difficult to provide valuable references and guidance for the further study and development in ship energy efficiency improvement. Therefore, the current progress made on the CFD-based energy-saving technology from the perspectives of energy consumption (hull optimization design, drag reduction technology), energy conversion (efficient propulsion devices), energy saving (energy-saving equipment), energy optimization (navigation state optimization), and the overall energy optimization (comprehensive performance optimization through CFD-based coupled dynamics analysis) is comprehensively analyzed in this research, as shown in Fig. 1. In addition, the challenges and problems of the CFD-based energy-saving technology are discussed, and the future research works are proposed, aiming to provide guidance for the development of ship energy-saving technologies. The contributions of this work mainly include the following aspects.

1) The ship energy-saving technologies based on CFD approaches have been discussed, including the hull optimization design, drag reduction technology, navigation state optimization, efficient propulsion devices, energy-saving equipment, and the overall performance optimization through the CFD-based coupled dynamics analysis.
2) A detailed summary of the progress on the CFD-based energy-saving technologies has been provided. On this basis, the challenges faced in the development of ship energy-saving techniques based on dynamics analysis methods are comprehensively analyzed, and the future research work on the CFD-based energy efficiency optimization is proposed.
3) This work can be regarded as an essential guidance for future research on the overall design optimization and fuel efficiency optimization of the ship based on CFD approaches, thereby promoting the development of the low-carbon shipping industry.

The other part of this research is structured as follows: the energy-saving performance of ship hull design optimization through CFD approaches is carried out in Section 2. Then, the energy-saving performance analysis of ship drag reduction based on CFD approaches is discussed in Section 3. In addition, the energy efficiency analysis of ship navigation state based on CFD approaches is conducted in Section 4. After that, the performance analysis of efficient propulsion devices based on CFD approaches is summarized in Section 5. Moreover, the performance analysis of energy-saving equipment based on CFD approaches is investigated in Section 6. Subsequently, the comprehensive performance optimization through CFD-based coupled dynamics analysis is illustrated in Section 7. Afterwards, the discussions on the energy-saving performance of the CFD-based technologies are carried out in Section 8. Finally, the challenges and prospects for the development of CFD-based energy-saving technology are outlined in Section 9.

2. Energy-saving performance of ship hull design optimization through CFD approaches

The ship hull design optimization aims to enhance the performance and operational efficiency of ships through the analysis and optimization of the hull shape, cross-section, and other aspects [30,31]. The detailed analysis of ship energy efficiency improvement methods based on CFD and the optimization design of the ship hull shape, cross-section, and bow/stern shape is carried out, aiming to provide theoretical and technical references for the energy-efficient ship hull design and optimization through CFD approaches [32].

2.1. *Energy efficiency analysis of the hull shape optimization*

The hull shape optimization refers to the adjustment and improvement of the ship's shape to enhance its performance, as shown in Fig. 2 [33]. The optimization of the hull shape considering factors, such as hydrodynamic resistance and wave resistance, can significantly reduce ship energy consumption and carbon emissions [34,35]. The numerical calculation based on CFD approaches can facilitate to obtain the

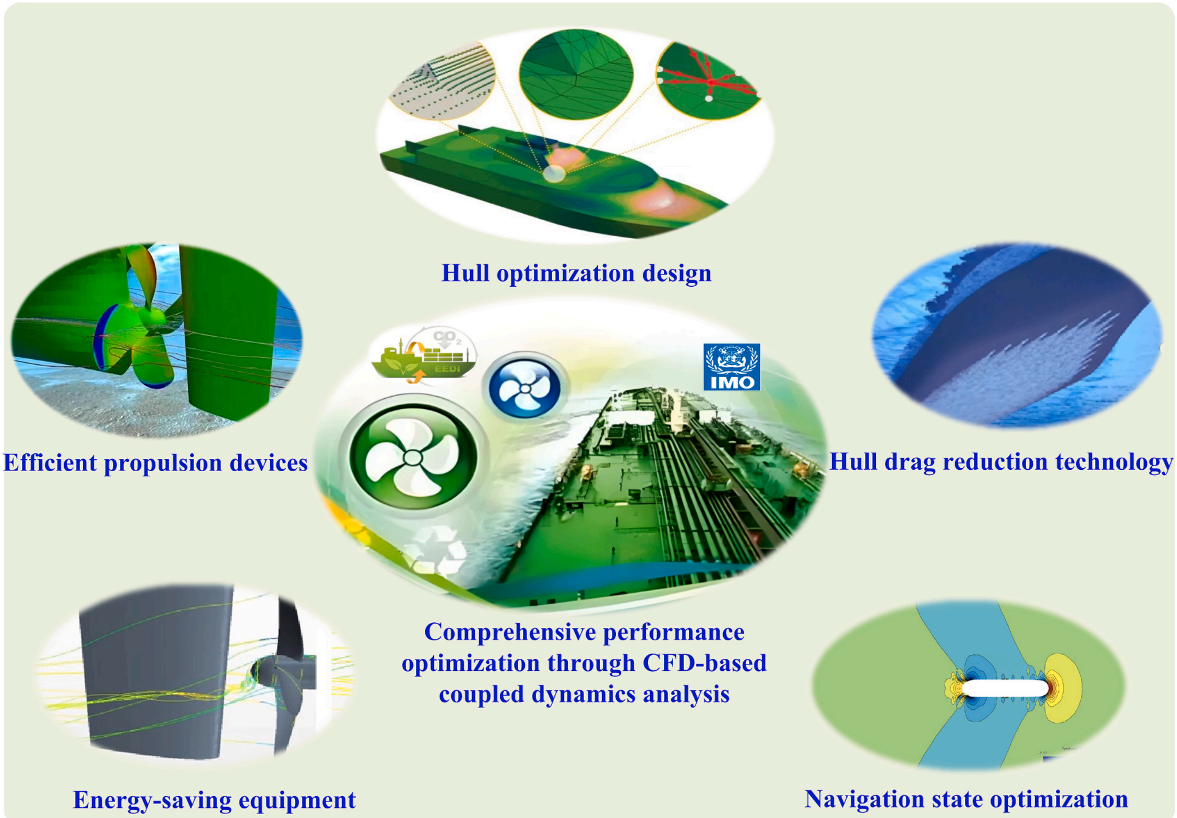


Fig. 1. Ship energy-saving technologies through CFD approaches.

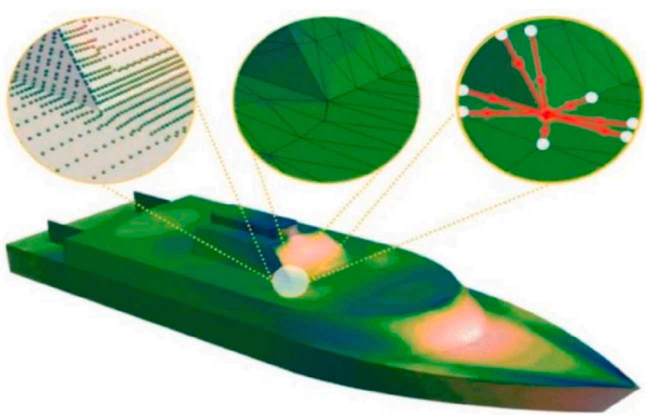


Fig. 2. Illustration of the hull shape optimization based on dynamics analysis [33].

influence of hull shape on the ship performance and resistance [36], and thus can achieve the optimized design of the hull shape to improve ship energy efficiency [37,38].

For the performance analysis of the hull shape, Wang et al. [39] proposed a non-uniform arch rib cylindrical ship hull, and the analysis results based on the Finite Element numerical simulation method and pressure experiment method showed that the structure had a smaller buoyancy coefficient and greater compressibility compared with traditional cylindrical ship hulls, which provides a new method and idea for further optimizing the ship hull shape and improving the ship fuel efficiency. In terms of the ship hull shape optimization methods, Zha et al. [40] proposed an improved maximum entropy (IME)-based hull hydrodynamic optimization method, which can optimize the hull shape to produce a smoother surface, and analyzed the dynamic characteristics and drag reduction effects based on CFD approaches. In addition, Kim and Yang [41] proposed a CFD-based shape optimization technique that can generate new ship hulls using basis functions, and adopted numerical simulation methods to analyze and verify the drag reduction effect by optimizing the hull shape through CFD approaches. Furthermore, Ouyang et al. [42] proposed an optimization process combining the Maximum Entropy Sampling method with CFD numerical simulation methods, which can reduce the total resistance of the hull by approximately 5.15 %. Wei et al. [43] proposed a novel hull shape optimization method based on Sampling-based Optimization (SBO) and CFD, and the study results showed that the hull shape optimization efficiency improves by at least 17.5 %. Cheng et al. [44] proposed a full-parameter hull lines optimization method based on CFD analysis, and the optimized hull resistance performance can be increased by 3 %. Nazemian and Ghadimi [45] optimized the trimaran hull through dynamics analysis based on the CFD method, which can reduce the ship resistance by 5.35 %. In addition, Nazemian and Ghadimi [46] proposed a shape optimization method based on CFD analysis. The numerical calculation results showed that the total resistance can be reduced by 6.67 % for a trimaran. Senov et al. [47] proposed a new method for in-detail hull form design based on the CFD and wave-based optimization, which can reduce the hull resistance by 8.9 %. Overall, the hull shape optimization based on dynamics analysis can effectively decrease the sailing resistance and thus the energy consumption of ships.

2.2. Energy efficiency analysis of the hull cross-section optimization

The cross-section of the hull refers to the section that is perpendicular to the longitudinal axis of the ship, and is usually used in ship design to define and describe important parameters such as the shape, size, and structural distribution of the ship [48,49]. During navigation, the ship would be subjected to various complex hydrodynamic loads, such as wave resistance and air resistance [50]. By using the CFD method combined with model experiments to optimize the design of the hull cross-section, and the shape and size of the hull cross-section, the frictional resistance and turbulence resistance can be reduced, and thus improving the overall energy efficiency of ships [51].

In recent years, the hull cross-section optimization has been carried out to improve the ship energy efficiency. Zong et al. [52] proposed a hull cross-section optimization method based on CFD and the Multi-island genetic algorithm (MIGA), and the obtained wave patterns of the hull are shown in Fig. 3. The study results showed that the total resistance coefficient of the ship can be reduced by 21.34 % compared with the original ship. Additionally, Miao et al. [53] adopted the free-form deformation (FFD) method to modify the surface of a catamaran and obtained a new hull shape. On this basis, they recommended a multi-dimensional optimization technique for enhancing the hydrodynamic performance of ships and conducted the resistance analysis using CFD approaches, as shown in Fig. 4, which can achieve good drag reduction effects. Cheng et al. [54] proposed an automatic hull surface correction technique combining CFD approaches with radial basis function interpolation optimization. The optimization results on the resistance performance of a 60-series ship showed that the correction technique can effectively improve ship energy efficiency.

2.3. Energy efficiency analysis of the bow shape optimization

The bow shape directly affects the ship sailing resistance, speed, and maneuverability [55]. The design optimization of the bow shape can improve the overall structure of the ship, reduce hull resistance, and enhance the overall energy efficiency of the ship [56,57]. The numerical calculation method based on dynamics analysis is an effective way to achieve the optimization of the bow shape [58]. Based on the dynamics analysis of different bow shapes by using the CFD method, the impact of the bow shape on the resistance and wave generation can be evaluated [59]. Liu et al. [60] compared the static water resistance of ships with and without bulbous bows, and found that the optimized bulbous bow can reduce the wave resistance coefficient, thereby reducing ship energy consumption. In addition, Yu et al. [61] adopted simulation-based design (SBD) and Particle Swarm Optimization (PSO) methods to analyze the hydrodynamic characteristics and resistance characteristics of different bow shapes under static water and irregular waves. The analysis results illustrated that the optimized bow shape can reduce wave resistance by about 9.3 %. Luo and Lan [62] used CFD approaches and Computer Aided Design (CAD) to achieve the optimal bow shape design and conducted hydrodynamic performance analysis of the hull, as shown in Fig. 5. The results showed that this method can effectively enhance the ship design efficiency and reduce wave resistance [63,64]. Additionally, Cheng et al. [65] adopted the CFD approach to optimize the bow of a container ship, and the numerical results showed that the CFD method has a good calculation accuracy and can effectively predict the variation trend of the ship's hydrodynamic performance. Moreover,

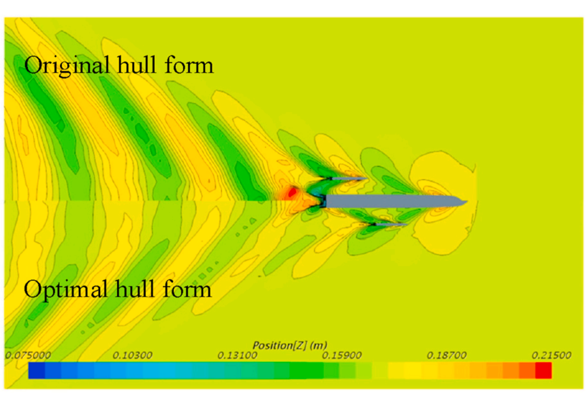


Fig. 3. Comparison of wave patterns [52].

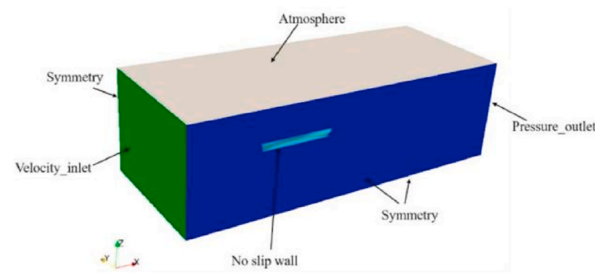


Fig. 4. S60 catamaran boundary conditions [53].

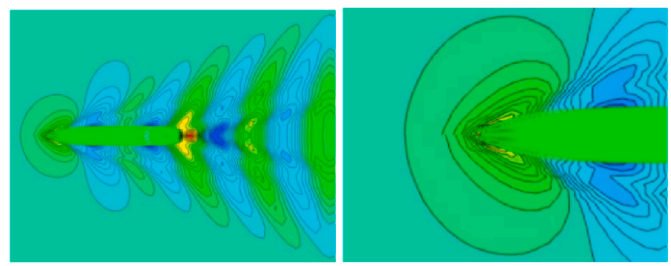


Fig. 5. Wave profile comparison [62].

Hong et al. [66] proposed a self-mixing method combining CFD and Design of Experiment (DOE) technology to optimize the bow shape of a fishing vessel, and conducted a hydrodynamic analysis of the bow, as shown in Fig. 6. The analysis results indicated that the resistance coefficient can be reduced by about 2 % by adopting the optimization method.

2.4. Energy efficiency analysis of the stern shape optimization

The optimization design of a ship's stern shape is one of the effective methods to decrease the ship navigation resistance [67]. By optimizing the design of the stern shape, the streamline properties of the flow field can be improved and the wave resistance can be reduced, and thus the overall ship energy efficiency can be enhanced [68]. Dynamics analysis methods can be used to simulate and calculate the hydrodynamic characteristics in the stern region, to effectively improve the hydrodynamic environment around the hull, reduce energy loss, and improve the ship energy efficiency [69,70]. Chen et al. [71] used the parametric modeling combined with the CFD approaches to optimize the contour design of the stern shape for a container ship, as shown in Fig. 7. The optimized stern flow field showed a significant decrease in turbulence, and the hull resistance can be reduced by about 9 %. In addition, Liu et al. [72] analyzed the interaction between the ship hull resistance and the stern flow field based on the CFD approach by optimizing the stern shape. The pressure distribution at the stern before and after

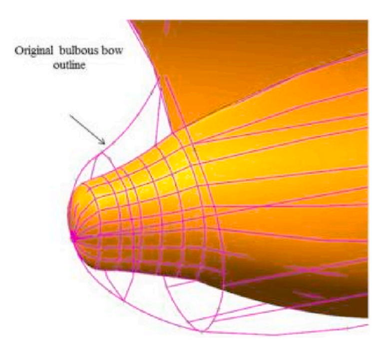


Fig. 6. 3D surfaces of the bow [66].

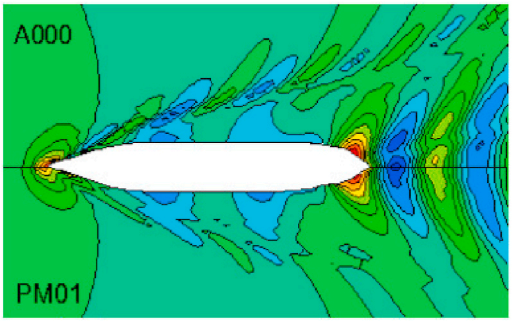


Fig. 7. Comparison of the CFD wake fields [71].

optimization is shown in Fig. 8. It is clear that the streamlined distribution of the hull after optimization is more uniform, and the total pressure resistance surround the stern is significantly reduced. Lu et al. [73] proposed a ship stern resistance optimization method based on the hull form transformation method and Response Surface Model (RSM) hydrodynamic solution method, which can reduce the hull resistance by about 7 % under the designed speed. Moreover, Hamed [74] proposed a multi-objective hydrodynamic optimization strategy for the stern design of a three-body ship using CAD combined with CFD calculations, which can reduce the hull resistance by about 13.3 % and increase the wake coefficient by 7.58 %.

2.5. Summary

The ship hull design optimization through the CFD approaches can improve the flow field and hydrodynamic characteristics around the ship, thereby reducing the sailing resistance and improving the overall energy efficiency of ships [75,76]. Although some studies on the hull design optimization based on the CFD approaches have been carried out, there are still some challenges that need to be addressed.

(1) The high-quality meshing and fine time steps for the CFD approach would result in high computational costs [77,78], which would consume more time and computation resources to achieve the hull optimization, especially under the coupling effects of various influencing factors.
(2) It is hard to obtain the experimental data to verify the effectiveness of the CFD-based ship hull design optimization method [79], and thus the accuracy of the CFD-based optimization results needs to be further investigated and demonstrated.

3. Energy-saving performance analysis of ship drag reduction based on CFD approaches

Ship drag reduction technology can reduce the sailing resistance by adopting the methods such as surface treatment, coatings, and CFD-based performance optimization, thereby reducing ship fuel consumption. A detailed analysis on the energy-saving performance of the drag reduction coating, and the drag reduction through bubble and air layer is carried out, aiming to provide a reference for the research and application of the ship drag reduction technology.

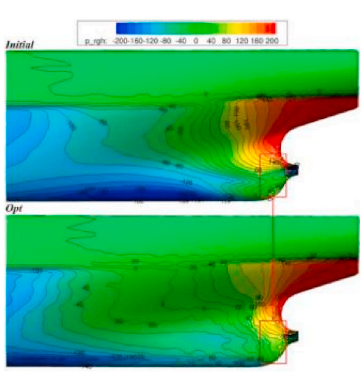


Fig. 8. Comparison of Stern pressure distribution [72].

3.1. Energy-saving performance analysis and optimization of drag reduction coatings

Ship coatings are used to protect the hull from corrosion, biofouling, and hydrodynamic losses caused by the marine environment [80,81], to reduce ship resistance and minimize the impact on the marine environment [82,83]. By using CFD-based approaches, the influence of different coatings on the fluid flow can be evaluated [84,85], and the ship resistance can be reduced by adopting suitable drag reduction coatings [86], thus improving the ship fuel efficiency [87,88]. Du et al. [89] established a model by adopting the dissipative particle dynamics (DPD) and CFD approaches to simulate and analyze the resistance-reduction efficiency of polymer coatings on the ship's surface, and verified the resistance-reduction performance of polymer coatings in external fluids, which can achieve the maximum drag reduction effect by 82.6 %. In addition, Kim et al. [90] proposed a low-friction surface coating with lubricant-injected spherical cavities for drag reduction, which can effectively reduce frictional resistance in turbulent high-speed flow. The drag reduction mechanism of the Marine Creature-Inspired Surface (MIS) method was studied by using CFD-based analysis methods, and the excellent resistance-reduction performance of the MIS method was validated through the application on the surface of high-speed flow ships. In addition, for the new types of drag reduction coatings (DRC), García et al. [91] studied the roughness characteristics of four different ship coatings under sea conditions and used CFD-based analysis methods to simulate and predict the effects of different coatings on the KCS hull resistance, as shown in Fig. 9. Moreover, Alza et al. [92] analyzed the influences of silicon-based coatings and traditional coatings on the drag reduction using numerical simulation methods and verified that the silicon-based coatings have better drag reduction performance, which is a solid foundation for the energy-efficient drag-reducing coatings development to further improve the ship energy efficiency.

3.2. Analysis of discrete bubble drag reduction technology

The drag reduction through discrete bubble (also called microbubble) can reduce the fluid resistance by introducing bubbles into the fluid [93]. With this technology, a two-phase flow in terms of gas-liquid with the flowing water underneath the ship could be generated as a lubricant for the ship's bottom, which can reduce the hull drag and energy consumption [94], as shown in Fig. 10. Yanuar et al. [95] proposed a new air lubrication device called the Winged Air Induction Pipe (WAIP) and calculated the effects of different hydrofoil clearance and angle of attack configurations on the total drag force and drag reduction by using the CFD analysis methods. The study results showed that the drag reduction effect with the obtained optimal parameters can be increased by 10 %. Yang et al. [96] calculated the effect of bubble drag reduction technology on the drag reduction effect of a river-sea bulk cargo based on CFD, and the research results showed that the total drag coefficient of the ship can be reduced by 19 %. Moreover, Yanuar et al. [97] conducted a comparative analysis of Microbubble Drag Reduction (MBDR) and Air-Layer Drag Reduction (ALDR) technologies, and the study results revealed that the MBDR has better performance than the ALDR technologies. In addition, in order to further explore the potential of bubble drag reduction for large ships, Mohammadpour et al. [98] proposed a gas-injected liquid lubrication system (GILLS) and analyzed the impact of GILLS on the drag reduction effect of the ship based on CFD approach. The results showed that the resistance of the hull can be

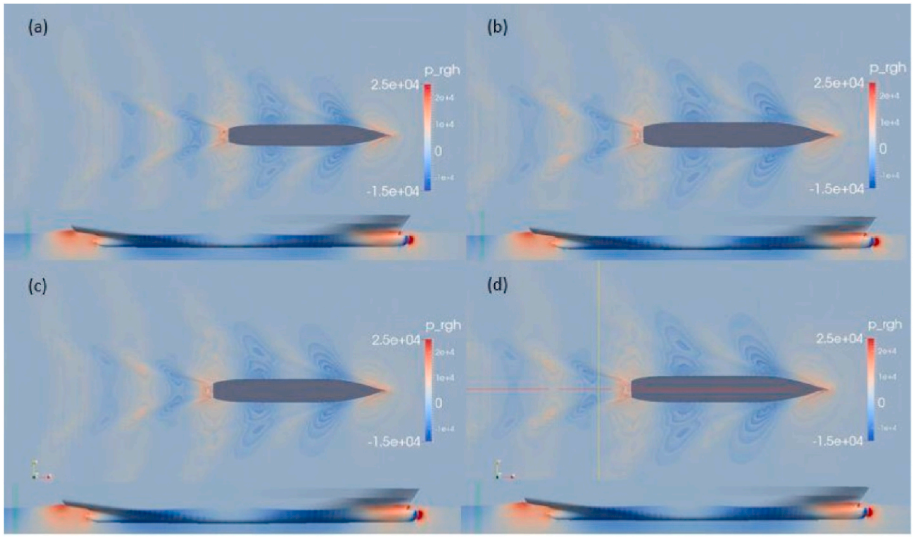


Fig. 9. Analysis on the dynamics performance of the hull with different coating materials [91].

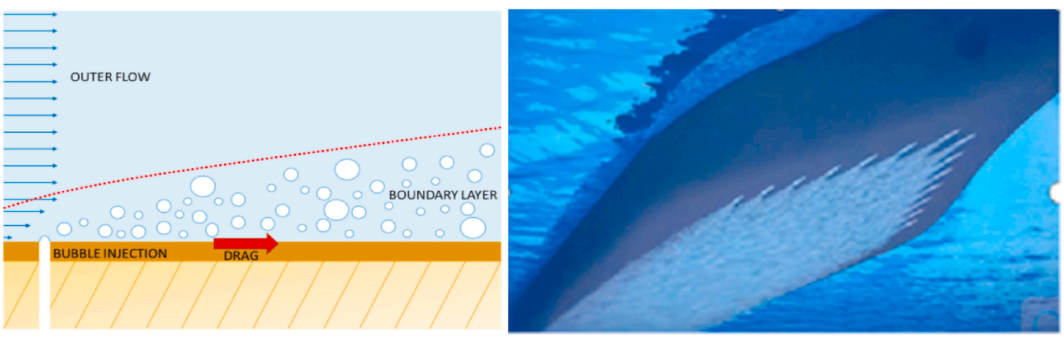


Fig. 10. Schematic diagram of bubble drag reduction technology [94].

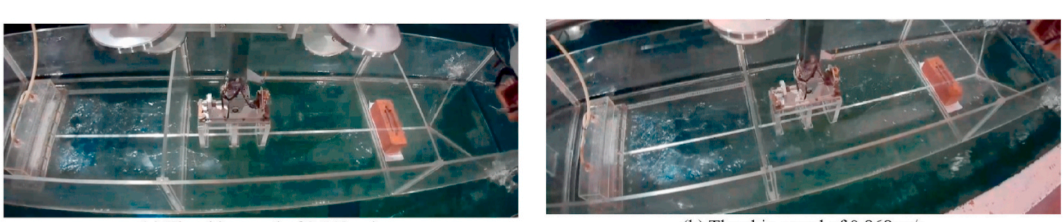
reduced by as much as 10.45 %.

(1) Influence of bubble size on the drag reduction effect

The bubble size is a significant factor affecting the resistance-reduction performance of the bubble drag reduction technology [99, 100]. Simulation and experiment analysis based on CFD analysis method (as shown in Fig. 11) indicate that smaller bubbles can achieve a better drag reduction effect, which is more conducive to enhancing the energy efficiency level of ships [101]. Giernalczyk and Kaminski [102] found that the size of bubbles in the MBDR region is a key factor that influences the resistance-reduction rate by adopting the CFD analysis method. The diameter of the bubbles significantly affects their distribution under the ship's hull and reduces the ship's resistance. At the same air inflow rate, larger water flow velocity produces smaller bubbles, and smaller bubbles are more conducive to improving the drag reduction effect. In addition, Moriguchi and Kato [103] analyzed the effect of microbubble diameter on the frictional drag under different incoming airflow velocities, and the study results showed that the increase of the mean void ratio can improve the resistance-reduction performance. This research finding has significant implications for improving the energy-saving effect of the bubble drag reduction.

(2) Influence of bubble size distribution on the drag reduction effect

The distribution of bubble sizes also influence the resistance-reduction performance of the bubble drag reduction technology, because the distribution status of bubble sizes affects the moving ways of the bubbles and the resistance in the liquid, and thus affecting the drag reduction effect. Zhao et al. [104] analyzed the influence of the bubble size distribution on the drag reduction effect through numerical simulation, as shown in Fig. 12. The study demonstrated that the size of bubbles gradually increases along the direction of free flow, with the largest bubble appearing behind the tail of the model. A significant



(a) The ship speed of 0.542 m/s (b) The ship speed of 0.868 m/s

Fig. 11. Experiment of the bubble morphology at two different ship speeds [101].

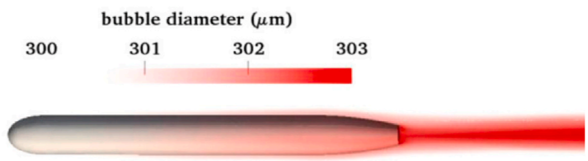


Fig. 12. Bubble diameter distribution analysis [104].

bubble merging effect occurs in the downstream area of the water stream. Therefore, the bubble-induced drag reduction performance can be improved to some extent by regulating the distribution status of bubbles reasonably.

(3) Influence of bubble void fraction on the drag reduction effect

Based on CFD approaches, studies have found that the void fraction has an obvious influence on the resistance-reduction effect in gas-solid two-phase flow [105]. Zhao and Zong [106] conducted an analysis of the drag reduction effect based on the Eulerian-Eulerian two-fluid model, the Volume of Fluid (VOF) model, and the Discrete Particle Model (DPM), the maximum drag reduction effect can reach by 7 %. The findings indicated that a higher air void fraction would obtain a better drag reduction effect. However, excessively high air void fractions may lead to excessive sparsity of the fluid, resulting in a decline in fluid flow performance, as shown in Fig. 13. Therefore, the reasonable adjustments are required in practical applications.

(4) Influence of bubble injection volume on the drag reduction effect

Bubble injection rate refers to the quantity of bubbles injected into the fluid during the application of the bubble drag reduction technology. Different bubble injection rates have varying effects on the drag reduction performance. Generally, a higher bubble injection rate can obtain a better drag reduction effect [107]. Tanaka et al. [108] found that a large number of bubbles generated by a bubble generator can inhibit the ship frictional resistance based on the CFD analysis method. Experimental data showed that skin-friction resistance of the ship model can be decreased by approximately 50 % after the bubble generator produces bubbles, and the resistance of the ship model will decrease with the increasing injection rate of bubbles. Therefore, the energy-saving performance of the bubble drag reduction technology can be enhanced by adjusting the bubble injection rate reasonably.

Above all, the study on the bubble-induced drag reduction technology is of significant importance in improving energy utilization efficiency, reducing emissions of pollutants, and enhancing ship fuel efficiency. The applications and drag reduction effect of the bubble drag reduction technology based on CFD approaches are presented in Table 1.

3.3. Analysis of continuous air layer drag reduction technology

The adoption of the continuous air layer drag reduction technology can decrease frictional resistance between gas and water surface by forming a layer of air film at the bottom of a ship, thereby reducing the drag force experienced by an object moving through a fluid [109,110]. A schematic diagram on the principle of the drag reduction through air layer is illustrated in Fig. 14 [111]. The drag reduction effect of microbubbles inserted into container hull model is studied based on CFD method, and the results show that the friction resistance of the ship can be reduced by as much as 27.6 %, and the drag reduction effect of microbubbles decreases with the increase of air flow [112].

Table 1
Application analysis of the bubble drag reduction technology based on CFD approaches.

Influencing factor	Application	Drag reduction effect	Reference
Bubble size	Air lubrication system (ALS)	The smaller the bubble, the better the drag reduction effect	[101–103]
Bubble size distribution	Optimization of bubble drag reduction	The drag reduction effect in downstream area is obvious	[104]
Bubble void ratio	Optimization of bubble drag reduction	The larger the void ratio of bubbles, the better the drag reduction effect	[106]
Bubble injection volume	Marine lubrication resistance optimization	Friction resistance can be reduced by about 50 %	[108]

In the practical applications of the drag reduction through air layer, the morphology of the ship's bottom surface and the method of air injection can be optimized through CFD numerical simulation methods to minimize frictional resistance and improve drag reduction effectiveness. Currently, the research and applications of the air layer drag reduction technology by adopting CFD approaches to enhance ship energy efficiency are shown in Table 2.

3.4. Summary

The energy-saving technologies, including coating [118], bubble [119], and air layer drag reduction technologies [120,121], have been widely studied and applied on ships, which can effectively reduce ship fuel consumption. The specific applications and the energy-saving effects of each technique are summarized, as show in Table 3. As it can be seen, various drag reduction technologies, including DRC, BDR, ALDR, have been employed to reduce ship energy consumption. The DRC technology mainly used in bulk carriers, container ships, oil tanker can effectively reduce the ship fuel consumption. In addition, the BDR technology is mainly used in hydrofoils, AUVs, cargo ships, which can reduce the ship fuel consumption by 5%–25 %. The ALDR technology, utilized in high-speed planning boats and container ships, can achieve a 4%–26 % reduction in ship resistance and can also decrease the ship sailing speed by 10%–30 %. Additionally, the WAIP have demonstrated a 16 % fuel reduction on a ferry. The ALDR applied to a typical Great Lakes ship can decrease net energy use by 10%–20 %. Moreover, the GILLS can reduce the hull resistance by 10.45 % on a Catamaran ROPAX ferry. These advancements in WAIP, GILLS, and ALS can further contribute to reducing resistance and fuel consumption of ships, thus advancing the low-carbon shipping industry.

Although the coating, bubble, and air layer drag reduction technologies can reduce ship resistance to some extent and have promising

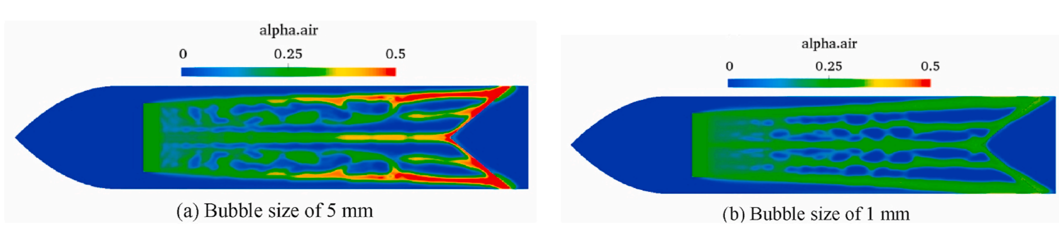


Fig. 13. The void distribution of different bubble sizes [106].

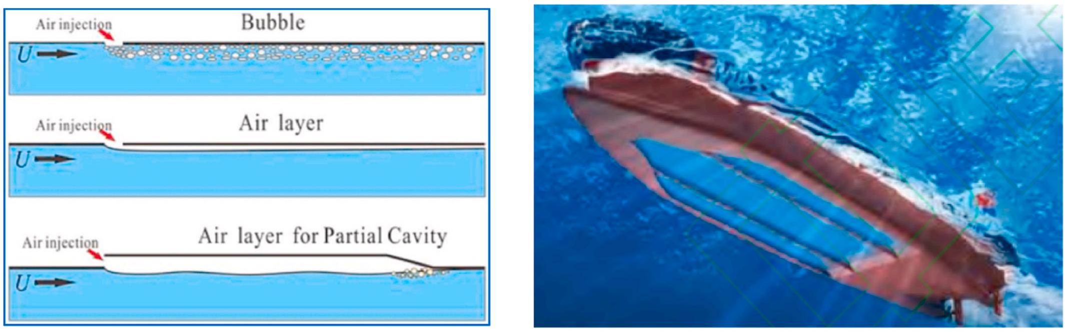


Fig. 14. A schematic diagram on the principle of the drag reduction through air layer [111].

Table 2
Research on the air layer drag reduction technology based on CFD approaches.

Considered factor	Application target	Drag reduction effect	Reference
Froude number, Drag coefficient	A glider with a cavity	The resistance can be effectively decreased	[113]
Hull speed	A planning boat without air injection	The drag can be decreased, and the speed can be effectively increased	[114]
Wave resistance, Froude number	A ship with an air chamber	The wave resistance can be reduced by 14 %	[115]
Drag reduction ratio	Model of a flat groove at the bottom of a ship	The drag reduction rate can be effectively increased	[116]
Sailing speed	High-speed planning boat	The sailing speed can be increased by 10%–30 %	[117]

application prospects [131,132], there are still the following challenges.

(1) Ship coatings would inevitably withstand long-term underwater environments, physical impact, and chemical corrosion, making durability of coatings very crucial [133]. In the future, the CFD should be used to simulate and predict the dynamics performance of coatings in complex sea conditions and develop more durable and long-lasting coating materials to enhance the overall ship energy efficiency.
(2) Currently, the bubble distribution study based on CFD analysis methods considering multiple factors is still in the preliminary stage, lacking accurate control of the bubble uniformity, size, and position. Therefore, further studies are needed to optimize the bubble control, thereby improving the energy-saving effect [134,135].
(3) Although drag reduction technology through air layer has been proven to effectively decrease the ship frictional resistance in theory [136], there is still a lack of field tests and verification in practical applications. Further studies and tests are required to verify the effectiveness and feasibility of the air layer drag reduction technology in ship fuel efficiency optimization [137].

4. Energy efficiency analysis of ship navigation state based on CFD approaches

The ship navigation state refers to the postures or motion states during navigation [138,139], and the optimization of which can reduce fuel consumption and environment footprint of the ship effectively [140,141]. The energy efficiency analysis under different ship navigation states based on dynamics methods [142,143] can lay a solid foundation for the optimization control of the ship navigation states [144].

Table 3
The applications of the drag reduction technologies.

Technology	Target ship	Energy-saving effect	Reference
BDR	Hydrofoil	The resistance can be decreased by 10 %	[95]
BDR	A river-sea bulk cargo	The total drag coefficient can be reduced by 19 %	[96]
GILLS	Catamaran ROPAX ferry	The resistance of the hull can be reduced by 10.45 %	[98]
BDR	Experimental ship model	The maximum drag reduction effect can reach by 7 %	[106]
ALDR	A Great Lakes ship	10 %–20 % of net energy savings can be achieved	[111]
ALDR	A container hull model	The maximum friction resistance can be reduced by 27.6 %	[112]
DRC	The towing tank VLCC model	The skin frictional drag can be deceased by 10 %	[122]
DRC	Ultra large container vessel Emma Maersk	It can save as much as 804749.4 kg of fuel for a voyage from Gdansk to Ningbo	[123]
DRC	Cruise ship Queen Mary II	It can save as much as 442027.1 kg of fuel for a voyage from Dubai to Southampton	[123]
DRC	A 176 k DWT bulk carrier	The fuel consumption can be saved by 11.7 %	[124]
DRC	A 176k bulk carrier	The fuel usage can be reduced by 48.06 %	[125]
BDR	A 120-m-long ship	The electricity can be reduced by 5 %	[126]
BDR	AUV	The resistance can be decreased by 25 %	[127]
WAIP	A fishing boat	The fuel usage can be reduced by 29 %	[128]
WAIP	A ferry	The fuel usage can be reduced by 16 %	[128]
WAIP	A cargo ship	The fuel usage can be reduced by 9.1 %	[128]
ALDR	A container ship	The resistance can be reduced by 4%–16 %	[129]
ALS	A 50,000 t medium range tanker model	The ship resistance can be reduced by 18.1 %	[130]

4.1. Analysis of the impact of trim on the ship fuel efficiency

The floating state of a ship has a significant impact on its stability and resistance during navigation [145]. Therefore, optimizing and controlling the ship's navigation state can effectively reduce sailing resistance and improve the energy efficiency level [146,147]. Ship trim refers to the difference in draft between the bow and stern of the ship [148,149], and an appropriate trim angle can reduce energy losses caused by external forces such as waves [150], thus improving the ship fuel efficiency [151]. The use of CFD-based methods can analyze the ship's resistance under different trims, which can provide an important foundation for the trim optimization [152,153]. Korkmaz et al. [154]

analyzed ship trim and drag reduction performance using EFD and CFD methods, which can effectively predict the optimal trim of the ship. Mahmoodi et al. [155] optimized the trim of a VLCC ship using the CFD method, and found that the ship's total resistance is the lowest at a bow trim of 0.2°, as shown in Fig. 15, which can achieve a reduction of 10.90 % in propeller thrust and 4.58 % in fuel consumption. Additionally, Fan et al. [156] achieved the trim optimization of a bulk carrier based on CFD analysis, which can reduce the ship resistance by as much as 4 %, and thus it is of great significance to decrease the energy consumption and carbon emissions of ships. Furthermore, Shivachev et al. [157] calculated the additional resistance of a ship under six different trim angles using a model-scale CFD method, and achieved the ship fuel efficiency optimization by obtaining the optimal trim angels. Additionally, Li et al. [158] conducted dynamic trim optimization research on a VLCC ship along typical routes using the boundary element method (BEM). Compared to the trim optimization results on the calm water surface, the dynamic trim optimization considering complex wind and wave conditions can achieve fuel saving of approximately 949.3 kg for a specific voyage, thus can effectively improve the ship energy efficiency.

In addition, ship draft refers to the effective immersion depth of a ship in water [159,160], which would affect the wetted surface area of a ship in water, directly influencing the stability and resistance of the ship [161,162]. Campbell et al. [163] calculated the ship resistance under different drafts and trim conditions using CFD-based method. The results indicate that increasing ship's draft under different speeds would result in the increased sailing resistance, which can be compensated by adjusting the ship's trim angle. At low sailing speeds, the resistance caused by the increased draft can be reduced by 10 % by the trim angle optimization.

4.2. Analysis of the impact of other navigation states on the ship fuel efficiency

The navigation state of a ship is influenced by the sailing resistance, including the water resistance and wind resistance [164]. The sailing resistance of a ship at different speeds can be obtained by using the CFD analysis methods, allowing for the decisions on the optimal sailing speed under various conditions to enhance the ship fuel efficiency. In addition, the sailing speed also affects the optimal trim of the ship [165]. Therefore, the joint optimization of the sailing speed and trim can further enhance the ship fuel efficiency. Li et al. [166] examined the effects of different speeds on the behavior of two ships sailing side by side under wave conditions through the CFD-based hydrodynamics calculations based on a Wigley III ship model, as shown in Fig. 16. Fan et al. [167] adopted the CFD and Dynamic programming (DP) algorithm to determine the optimal speed for different segments of a ship's voyage. Meanwhile, the optimal trim was also obtained under the optimized speeds. The proposed method can reduce the total power consumption by 7.64 %, which is of significant importance for further improving the ship energy efficiency.

4.3. Summary

In summary, some studies have been conducted on the energy efficiency optimization through ship navigation state based on CFD approaches, which can lay solid foundations for the ship energy efficiency optimization. However, there are still some problems and limitations that need to be addressed.

(1) The comprehensive modeling and analysis methods considering the complex and uncertain dynamic factors, such as wind, waves, and currents, have not yet been established for ship navigation state optimization.
(2) The research on the fuel efficiency improvement by ship navigation state optimization based on dynamics analysis methods often involves certain simplifications and assumptions, which would affect the effectiveness of the energy efficiency optimization method. Thus, it is vital to validate the fuel efficiency enhancement effects of the navigation state optimization under different environmental and operational conditions through specific case studies.

5. Performance analysis of efficient propulsion devices based on CFD approaches

Dynamics analysis methods can be used to achieve the performance analysis and the design optimization of the propulsion systems, including propellers, rim thrusters, water jet propulsion, and podded propulsion unites, which can enhance the propulsion efficiency and improve the overall energy efficiency of the ship. In addition, optimizing the combustion performance of marine diesel engines can also effectively improve fuel utilization and reduce fuel consumption based on CFD [168,169], thus promoting the green development of ships [170,171].

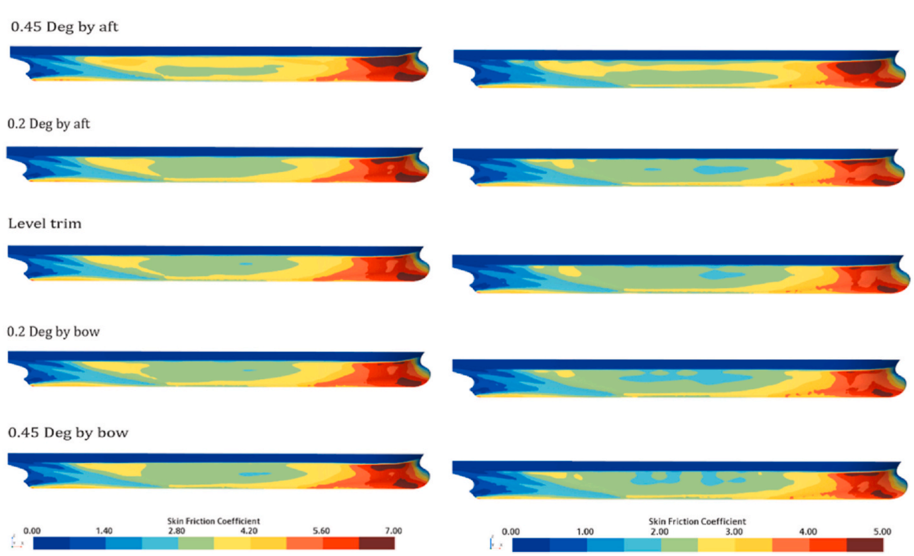


Fig. 15. Skin friction coefficient distribution on KVLCC2 hull [155].

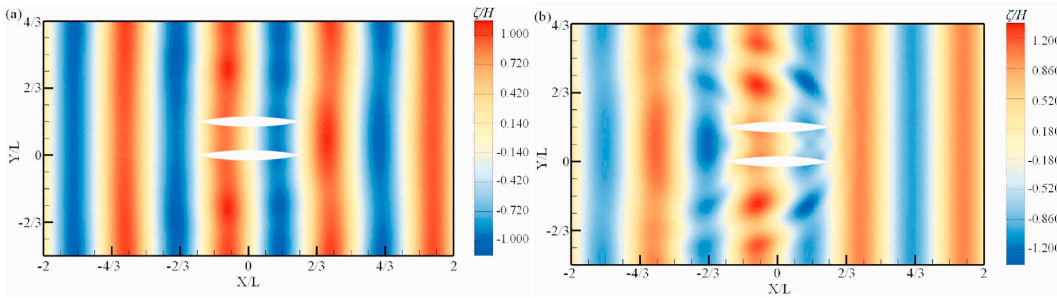


Fig. 16. The wave field distribution with different values of Fn [166].

5.1. Optimization analysis of the propeller performance

The propeller, as the important device for propulsion [172], has a specific effect on the maneuverability, stability, and energy efficiency of the ship [173]. The hydrodynamic properties analysis of propellers by CFD approaches can provide insights into the relationship between the dynamic parameters, such as thrust, resistance, and torque, and can achieve the optimized design of propellers with significant implications for improving ship energy efficiency [174]. The open-water performance of a propeller plays a crucial role in the ship propulsion performance [175]. Eom et al. [176,177] analyzed the relationship between the open-water performance and the ratio of immersion depth to diameter, as well as the advance coefficient of a propeller based on CFD approaches. They also analyzed the influence of immersion depth and skew angle on the open-water characteristics of the propeller under the wave conditions, and obtained the trend of propeller performance varying with the skew angle. In addition, Liu et al. [178] optimized the propeller performance based on CFD approach, which can improve the propeller wake performance by 2 %. Stan et al. [179] proposed a novel backflow marine propeller and optimized the propeller through CFD approach, which can improve the efficiency of the propeller by 8 %–9 %. Liu et al. [180] calculated the ship resistance, propeller open-water performance, and other parameters by adopting the CFD analysis method, as shown in Fig. 17. On this basis, the analysis and prediction of the power performance at different ship speeds were conducted, laying a solid foundation for improving the operational performance of propulsion systems. Lovibond et al. [181] analyzed the hydrodynamic performance of the propeller based on CFD, and found that the ship efficiency can be improved by 62 % at lower propeller thrust and torque coefficients. Furthermore, the combined propellers have high propulsion efficiency and good energy-saving effect. Su et al. [182] analyzed the hydrodynamic performance of single propeller and the combined propeller based on CFD. The results show that the open-water efficiency of the combined propeller is significantly higher than that of the single propeller under the same power, and the combined propellers have high propulsion efficiency and good energy-saving effect.

The dynamic characteristics analysis of full-scale propellers can avoid the problem of analysis errors and uncertainties caused by using the model scales, and can improve the accuracy and effectiveness of the performance analysis [183]. Kim et al. [184] used CFD-based analysis method to predict the impact on different wall y + values on the dynamic performance of a full-scale propeller in real and virtual fluids, as shown in Fig. 18, which is of great importance for promoting the research and real-world applications of the propeller performance optimization.

5.1.1. Performance analysis of twin-propeller propulsion systems

Twin-screw propulsion systems can reduce hydrodynamic losses and improve propulsion efficiency by utilizing the interaction between two propellers, thereby enhancing the energy efficiency of ships. Compared to single-screw propeller, twin-screw systems can provide greater thrust and achieve higher speeds with the same power input [185,186]. Lu et al. [187] proposed a twin-propeller twin-rudder system (TPTR) and analyzed the hydrodynamic performance by using CFD analysis, and also conducted an analysis of the ship maneuverability with the aid of CFD analysis. Vimala et al. [188] obtained joint computational results of pressure, velocity, and turbulence under propeller operating conditions using the FLUENT software. On this basis, the optimal propeller rotational speed was determined, which can effectively improve the propeller propulsion efficiency. Additionally, Acanfora et al. [189] performed CFD simulations to analyze the dynamics performance of twin-screw propellers under complex irregular sea conditions, which is of significance for improving the fuel efficiency and reducing pollution gas emissions. Moreover, Guo and Zou [190] developed a numerical model for twin-screw propeller thrust and torque based on the CFD method, as shown in Fig. 19. On this basis, the hydrodynamic characteristics of the propeller were investigated and the variation law of propeller thrust load during ship turning motion was obtained, which is significant for the energy efficiency prediction of the twin-screw ships.

5.1.2. Energy efficiency analysis of counter-rotating propellers

Counter-rotating propellers (CRP) is a propulsion system consisting of two propellers rotating in opposite directions. In the CRP system, the rear propeller compensates for the energy losses caused by the front propeller, thereby improving ship propulsion efficiency [191]. Numerical simulations of CRP dynamics can be conducted using CFD techniques to quantitatively analyze the propulsion efficiency and performance of the CRP system, thus enhancing the ship energy

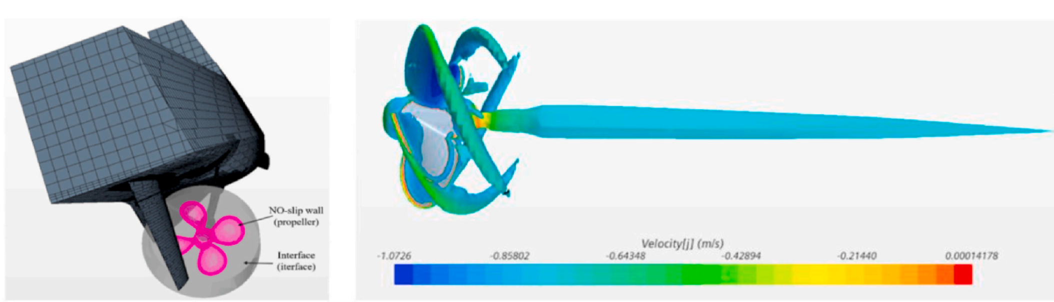


Fig. 17. Propeller open-water performance analysis [161].

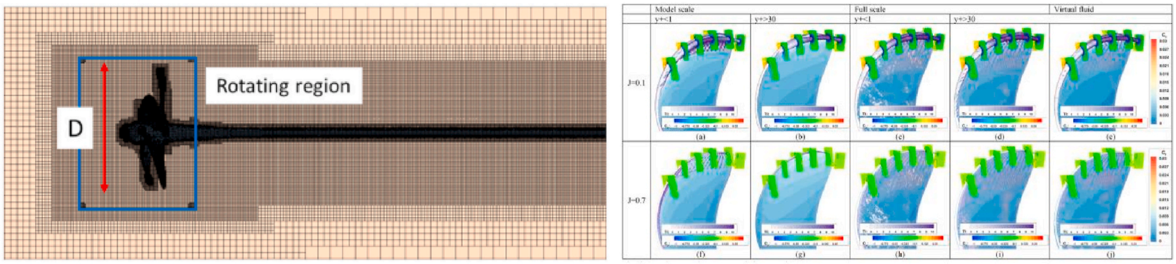


Fig. 18. Analysis of the dynamic characteristics of full-scale propellers [184].

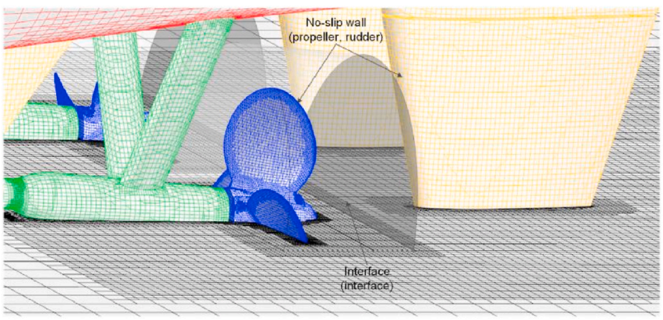


Fig. 19. The CFD computational modeling of the twin-propeller propulsion systems [190].

efficiency. Pereira et al. [192] analyzed the hydrodynamic characteristics of CRP wake under different operating conditions using dynamic particle image velocimetry. On this basis, the influence of the relative position of CRP propellers on flow characteristics was obtained. Capone et al. [193] studied the hydrodynamic characteristics of CRP systems under three different thrust coefficients and analyzed the interaction between the vortex systems generated by the front and rear propellers, as shown in Fig. 20. The results emphasized the importance of hydrodynamics analysis of wake flow for improving the energy efficiency of CRP propulsion systems. In terms of CRP optimization design, Grassi et al. [194] proposed a design method based on the hydrodynamics analysis to obtain the optimal shape of the CRP system. The experimental results demonstrated that the CRP system has a higher flexibility. In addition, Nowrouz et al. [195] conducted an optimization analysis of the CRP system using the CFD analysis method, and the hydrodynamics performance of the CRP system was simulated and calculated, as shown in Fig. 21. On this basis, an optimization model for CRP by combining the genetic algorithm and Kriging method was constructed, which is of great significance for optimizing marine propellers and thus improving the energy efficiency of ships.

5.1.3. Energy efficiency analysis of variable pitch propellers

Variable pitch propellers (VPP) can improve efficiency under different sailing conditions by changing the blade angle of the propeller [196], thereby improving ship energy efficiency [197]. The performance of VPP is closely related to the flow field. The CFD analysis methods can be used to precisely simulate the flow field around VPP and optimize the related parameters, such as blade shape and angle, to obtain the performance characteristics of different design schemes. Zhu et al. [198] analyzed the dynamic performance of the VPP and traditional propellers under different operating conditions by using CFD methods, as shown in Fig. 22. In addition, Gypa et al. [199] optimized the performance of VPP based on the CFD analysis method to minimize the total power consumption of the ship. The design and optimization of VPP under high load conditions can yield solutions with the lowest total energy consumption. Ma and Wang [200] carried out a comparison between the VPP and Fixed-Pitch Propellers (FPP) by calculating thrust and torque using the lattice Boltzmann method. The study results showed that the VPP can effectively improve the ship sailing speed and propeller efficiency, and can make up for the efficiency error caused by the FPP design, which is significant for the drag reduction and energy efficiency improvement of ships.

5.1.4. Performance analysis of other types of propellers

Ducted propeller is an advanced propulsion system widely used in the optimization of propeller efficiency [201]. An et al. [202] used a

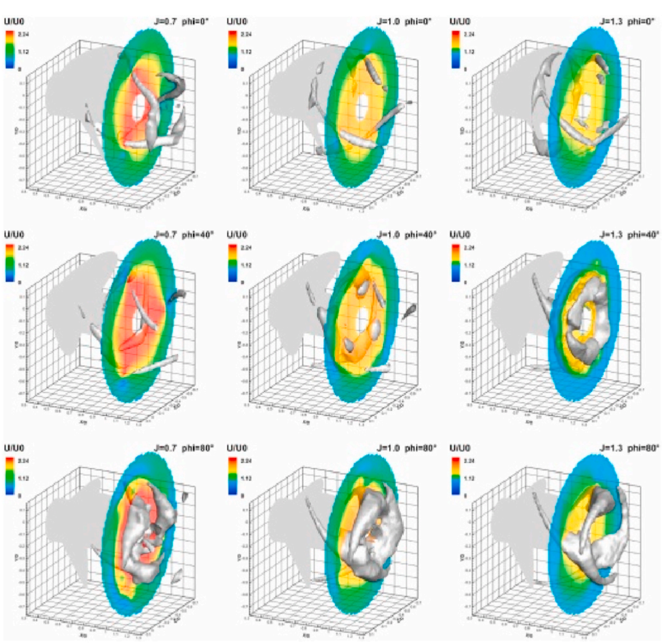


Fig. 20. The mean and normalized velocity field [193].

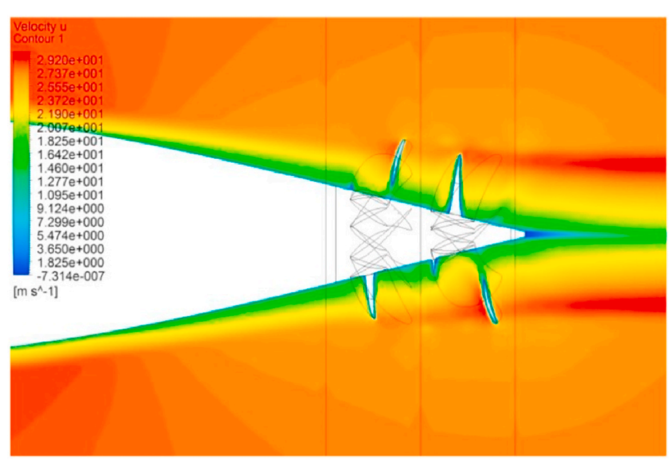


Fig. 21. The axial velocity contour [195].

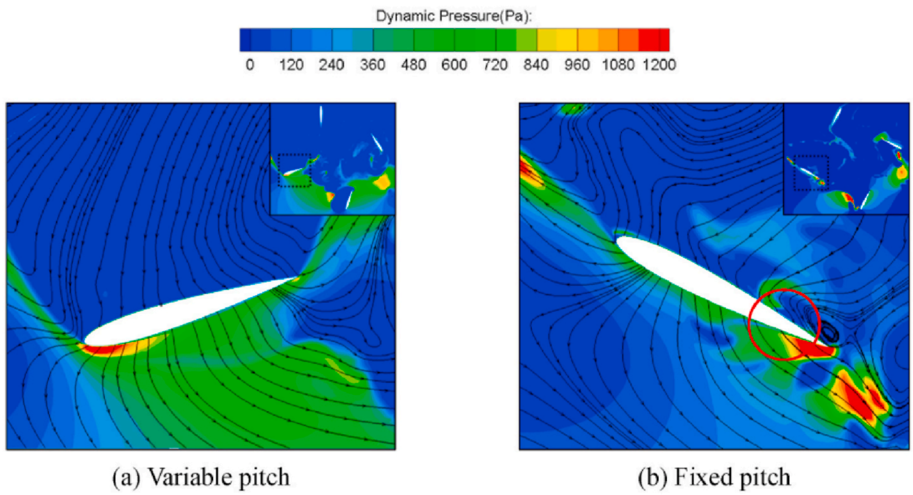


Fig. 22. The pressure distribution of the VPP and FPP [198].

bi-directional fluid-structure coupling method to analyze the influence of the tip gap between composite metal ducted propellers and blades on the hydrodynamics performance and pressure fluctuations of the propeller, as shown in Fig. 23. The results showed that using composite materials with smaller tip gaps can achieve better hydrodynamics performance of the propeller and can significantly reduce pressure fluctuations on the blade surface. In addition, Joung et al. [203] focused on the shape optimization of an AUV with a ducted propeller based on CFD approach. The optimization results showed that the resistance of the AUV is the lowest when the angle of attack of the propulsion nozzle is 9.15°. Furthermore, Li and Sun [204] carried out numerical simulations focusing on a helical propeller's 3D model by adopting the CFD analysis method, and analyzed the correlation between the dynamics parameters and scale effects. The results showed that the thrust and torque coefficients would increase with the Reynolds number within a certain range, which can lay a foundation for the development and practical applications of the propellers.

5.2. Optimization analysis of the rim-driven thruster performance

Rim-driven thruster is a new kind of ship propulsion system [205, 206], which is mainly driven by an electric motor and gearbox to generate thrust and propel the ship forward [207]. By directly mounting the propeller blades on the rim, the transmission system found in traditional thrusters can be eliminated, which can reduce the energy losses effectively [208]. For the dynamics parameters optimization of the rim-driven thrusters, Jiang et al. [209] established three novel hydrodynamics simulation models for Rim-driven Counter-Rotating Thrusters (RD CRT) by adopting the CFD analysis method. They simulated the frictional power loss and flow characteristics in the gap channels, and investigated the effects of the gap on the hydrodynamic performance of the thruster through CFD analysis, as shown in Fig. 24. The study suggested that the RD CRT has a better thrust coefficient, torque coefficient, and maximum efficiency compared to a single propeller. Zhai et al. [210] analyzed the influence of optimized duct on the rim-driven thruster propulsion performance based on CFD. The results showed that the duct optimization can improve the rim-driven thruster propulsion efficiency by 3.3 %. In addition, Cai et al. [211] analyzed the open-water performance of rim-driven thruster to improve the efficiency of rim-driven thruster based on CFD, and the results showed that the improved rim-driven thruster can improve the open-water efficiency by 10.9 %.

Moreover, Cao et al. [212] predicted the wake field and load distribution of a rim-driven propulsion device using the CFD method, and analyzed the dynamics characteristics of the propeller blade based on the prediction results, and established a numerical method that can better predict the hydrodynamics performance of the rim-driven thruster. Yang [213] investigated the effects of hydrodynamics parameter on the propeller by adopting numerical simulation. On this basis, the power consumption and heat dissipation issues of the shaftless rim thruster and the hydrodynamics performance of the propeller are analyzed. The obtained relationship between blade thickness and friction torque on the inner and outer surfaces of the rim-driven device is an important foundation for further optimizing the performance and efficiency of the thruster.

5.3. Optimization analysis of the water jet propulsion performance

The water jet propulsion system can propel the ship forward through the generated powerful thrust by utilizing the reaction force of water [214]. The advantages of water jet propulsion systems for ships include good maneuverability, quick response, simple operation, and low noise

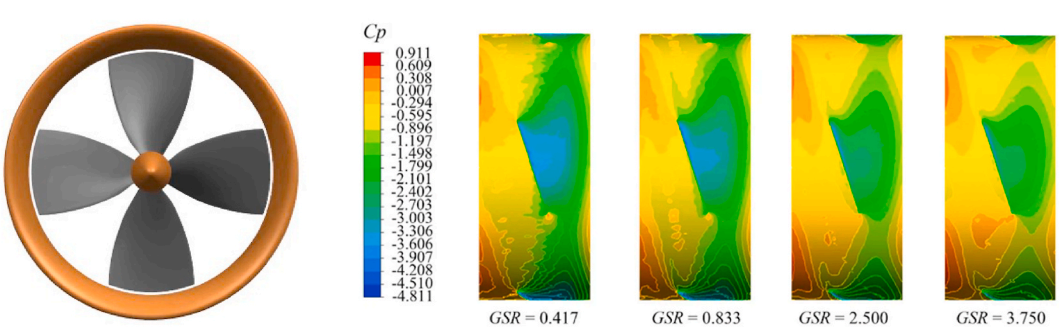


Fig. 23. Pressure distribution of the composite ducted propellers [202].

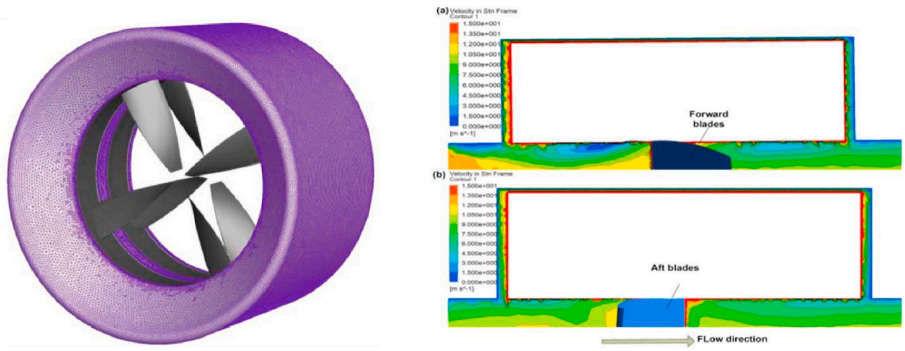


Fig. 24. Mesh division and gap fluid velocity distribution of the rim-driven thruster [209].

[215]. However, there are also limitations associated with water jet propulsion systems, such as high energy consumption and the dependence on water flow [216]. Therefore, optimizing the hydrodynamics parameters of water jet thrusters through CFD analysis, and meanwhile, considering factors, such as ship motion states and characteristics of the water jet propulsion system, can help to achieve the appropriate design and application strategies, thereby enhancing the ship fuel efficiency.

In the aspects of numerical simulation analysis, Zhao et al. [217] utilized CFD simulations to analyze the fluid dynamics and flow properties of water jet propulsion systems during the mooring condition. On this basis, the open-water characteristics of water jet propellers, and the hydrodynamics of the water jet-propelled ships were obtained. This study provided an effective method for the hydrodynamic performance analysis of the water jet propeller, which can contribute to the optimization of ship energy consumption. Eslamdoost et al. [218] conducted CFD-based research on the pressure variation of the ship hull caused by the water jet system under static and dynamic water conditions, as well as the variation of frictional resistance, obtaining the resistance variation process within a range of speeds. Additionally, Eslamdoost et al. [219] suggested a pressure jump method that can effectively predict the flow rate of water jet propulsion systems. The interaction between thrust and hull frictional resistance of water jet propulsion systems was investigated by using the potential flow/boundary layer theory and CFD method, as shown in Fig. 25. Moreover, Liu et al. [220] established a double-waterfoil ship model equipped with a water jet propulsion system. On this foundation, the effects of water jet propulsion systems on the performance (e.g. resistance) of the stern water foils were simulated and predicted by using the CFD analysis methods, which can provide important guidance to improve the hydrodynamics performance of ships. Shirazi et al. [221] analyzed the propulsion performance when the water jet propulsion device was installed at the back of the hull, which can improve the performance of the entire hull by 82 %. Additionally, Lee et al. [222] optimized the shape of the slit that injects the jet from the surface of the propeller based on CFD approach, improving the propeller propulsion efficiency by 2 %, which lays a foundation for the performance optimization of the ship propeller.

5.4. Optimization analysis of the podded propulsion performance

Podded propulsion systems are the devices directly installed on the bottom of ships or underwater vehicles [223,224]. Podded propulsion systems can rotate 360°, providing omnidirectional thrust, and making ships more maneuverable during operation and navigation. They can also effectively reduce vibration and noise generation [225,226]. The use of CFD-based analysis methods allows for a quantitative evaluation of the energy efficiency performance of podded propulsion systems during ship operation, which can facilitate to obtain the optimal dynamics performance and provide strong support for the optimization of podded propulsion systems, thus enabling ships to improve energy efficiency under various operational conditions. Zhang et al. [227] studied the unsteady dynamics characteristics of the CPR-POD mixed propeller propulsion system and calculated the load distributions of two propellers by adopting the CFD method, to reveal the characteristics of the unsteady flow field. In addition, Wu et al. [228] analyzed the influence of installation position on ship propulsion efficiency. The research showed that when the podded propulsion is in the initial position, the maximum propulsion efficiency of the podded propulsion cruise ship is 0.7010 at the initial installation position. Additionally, Park et al. [229] suggested an innovative approach for predicting the full-scale performance of podded propulsion systems, and provided a correction method for the thrust and torque of podded propulsion systems by comparison analysis between the numerical simulation results and model tests. Moreover, Choi et al. [230] conducted dynamics simulation calculations of podded propulsion systems using CFD methods from model to full scale, and analyzed the effects of the Reynolds number scale and load on

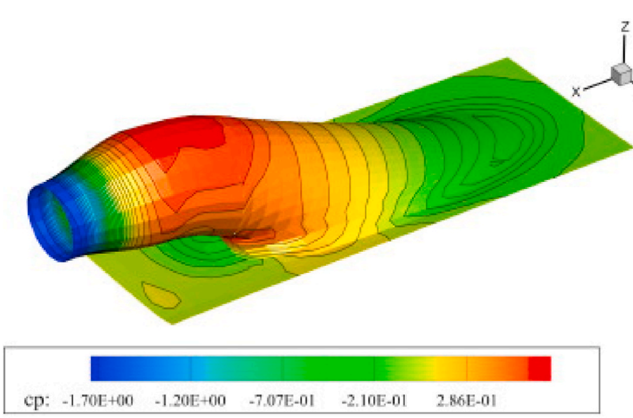


Fig. 25. Pressure distribution diagram of water jet propulsion system [219].

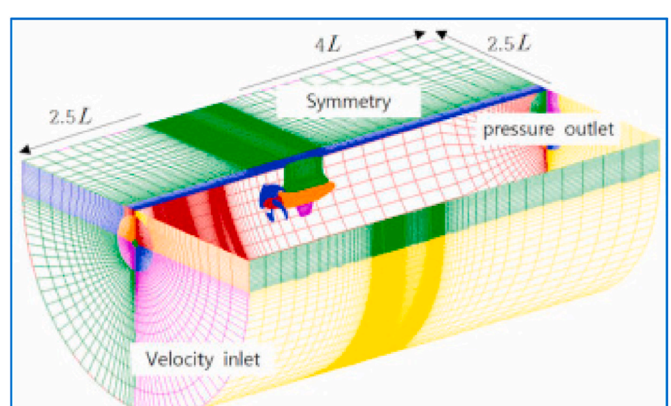


Fig. 26. The CFD meshing of the podded propulsion system [230].

the interaction between propeller blades and the pod housing, as shown in Fig. 26. The simulated outcomes closely matched the experimental results, indicating the feasibility of using CFD methods to predict the hydrodynamics performance of the podded propulsion systems.

5.5. Summary

The performance analysis and optimization design of efficient propulsion systems are important aspects of enhancing ship energy efficiency. The design optimization and performance improvement of the commonly used propellers, twin propellers, rim-driven thrusters, water jet propulsion systems, podded propulsion systems based on CFD approaches can effectively decrease the ship fuel consumption and greenhouse gas emissions. In addition, the combustion performance of marine diesel engines can be also optimized based on CFD, thus enhancing the ship energy efficiency [231,232]. The research status on the efficient propulsion system optimization based on CFD approaches is shown in Table 4.

Although a considerable numerous studies have been conducted on the performance optimization of efficient propulsion systems based on dynamics analysis, there are still the following issues and challenges.

(1) The dynamics analysis of efficient propulsion systems for ships does not fully consider the influence of various coupled factors, such as complex operational conditions, payloads, and speeds [233].
(2) There is still a lack of comprehensive optimization analysis for parameters, such as blade shape, pitch distribution, and roughness of the propellers.
(3) The hydrodynamics losses of propellers during the dynamics analysis are overlooked, and there is a lack of practical operational data validation for the complex hydrodynamic performance analysis of the efficient propulsion systems.

6. Performance analysis of energy-saving equipment based on CFD approaches

Ship energy-saving equipment can reduce the sailing resistance and improve the propulsion efficiency of ships, which is important to reduce the ship energy consumption and CO_2 emissions [234,235]. The commonly used ship energy-saving equipment mainly include the pre-propeller energy-saving devices (such as flow rectifying ducts and fins) and post-propeller energy-saving devices (such as rudders, bulbous bows, and stern appendage fins), as well as the wind-assisted propulsion systems, etc. The dynamics analysis and optimization of the energy-saving appendages is significant to reduce energy consumption [236,237], thereby improving the fuel efficiency of ships [238,239].

Table 4
Research on the efficient propulsion system optimization based on CFD approaches.

Types	Reference	Method	Optimization parameter	Effect
Ordinary propeller	[176–182]	CFD	Hydrodynamics performance, and open-water performance	The resistance is reduced and the propulsion efficiency is improved
Twin propeller	[187–190]	CFD	Propulsion efficiency	Propeller driving efficiency is improved
Rim propeller	[210–213]	CFD	Hydrodynamics performance	Torque loss is reduced and efficiency is improved
Water jet propulsion plant	[217–222]	CFD	Hydrodynamics parameter	The drag reduction rate is as high as 25.7 %
Pod propeller	[288-230]	CFD	Propeller dynamics parameter	Resistance is reduced and propulsion efficiency is improved

6.1. Performance analysis of energy-saving appendages

By optimizing the performance of energy-saving devices such as rudder bulb thrust fins, stern ducts, and pre-swirl stators (PSS) using CFD methods, the overall hydrodynamic performance of the ship can be improved, which is highly significant for improving the energy efficiency of ships. Shen et al. [240] used CFD numerical simulation to analyze and predict the scale effect of ship propulsion efficiency with rudder bulb thrust fins, as shown in Fig. 27. The model-scale simulation presented a 4.85 % improvement in ship propulsion efficiency, while the full-scale simulation showed a 2.28 % improvement in ship propulsion efficiency. The research results can help to better understand the mechanism how the energy-saving appendages boost the energy performance of ship. Wu et al. [241] used CFD simulation to predict the influence of stern ducts on the overall resistance and viscous flow field around the ship. It was found that the ship can achieve the most effective drag reduction effect when the angle of the stern duct is 7°, which can reduce the ship resistance by approximately 2.49 %. In addition, Bakica et al. [242] studied the impact of different propulsion methods on the hydrodynamic performance of pre-swirl stators (PSS) using CFD and Finite Element Method (FEM), and found that the PSS can effectively improve the efficiency of ship propellers by approximately 4.69 %. Furthermore, Obwogi et al. [243] optimized the hydrodynamic parameters of the rudder-bulb-fins system using CFD theory and numerical simulations and achieved the combined optimization of the rudder-bulb-fins system through CFD-based dynamics analysis for ship fuel efficiency improvement. The analysis results showed that the rudder-bulb-fins system can improve propeller efficiency by 2.63 % under optimal conditions.

6.2. Performance analysis of other energy-saving equipment

In addition to the rudder bulb thrust fins, stern ducts, and PSS, the performance analysis and optimization of other energy-saving equipment has also been investigated [244,245], such as the Gate Rudder System (GRS) [246,247], hull vane [248,249], Pre-duct [250], Mewis Duct [251], wing-typed sail [252,253], and Flettner rotor [254,255]. The CFD approaches can be employed to optimize the rudder angle of GRS [256] and the shape of the sails [257,258] to improve the dynamics performance of those equipment [259,260], thereby decreasing the energy usage and CO_2 emissions of ships [261]. Kiryanto et al. [262] analyzed the effect of adding hull vane on ship drag reduction based on CFD, which can reduce the total resistance of the ship by about 20 % compared with that of the ship without hull vane. In addition, Soma and Vijayakumar [263] analyzed the effect of hull vane on ship resistance based on CFD, as shown in Fig. 28. The results showed that the total drag coefficient of the ship equipped with hull vane can be reduced by 7 % compared with the original hull. Atlar et al. [264] analyzed the energy-saving effect of GRS on ships based on CFD. The results showed that the GRS can achieve energy saving by 10 %. Munazid et al. [265] analyzed the influence of Pre-duct on the propulsion performance based on CFD approach. The results showed that the use of Pre-duct can improve the ship propulsion performance by 3 % for fishing boats, which can effectively enhance the ship energy efficiency. Trimulyono et al. [266] analyzed the influence of Mewis Duct on the propeller performance based on CFD approach. The results showed that the use of Mewis Duct can improve the propeller thrust by 3–5%. In addition, Shen et al. [267] proposed a new partial duct and unconventional pre-swirl fin combination system and analyzed the energy-saving effect of the combined system on ships based on CFD, which can achieve energy

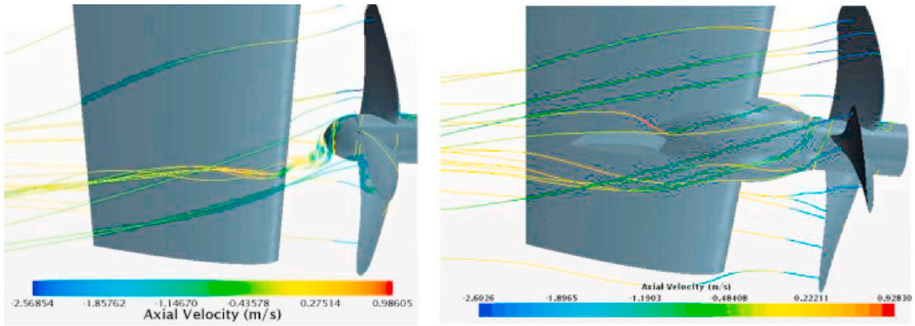


Fig. 27. Dynamic characteristics analysis of thrust fins of rudder ball based on CFD [240].

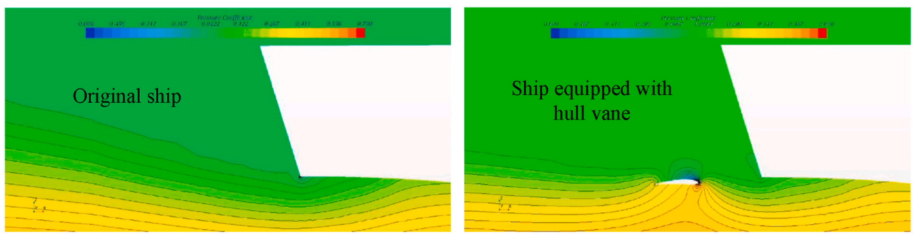


Fig. 28. Comparison of the ship pressure coefficient based on CFD analysis [263].

saving by as much as 4.26 %. Tan et al. [268] analyzed the influence of the propeller boss-cap-fin (PBCF) on ship propulsion system performance based on CFD. The results showed that the use of PBCF can improve the ship propulsion performance by 3 %, thus enhancing the ship energy efficiency. Additionally, Zhang et al. [269] conducted the analysis and optimization of the dynamics performance of wing-typed sails by using CFD combined with the PSO algorithm, which can increase the system's thrust coefficient by 6.5 %.

6.3. Summary

The research and applications of energy-saving appendages for ships based on CFD approaches can reduce carbon emissions effectively [270, 271], and contribute to the sustainable development of the shipping industry [272,273]. The research status on the performance optimization of energy-saving equipment based on CFD approaches is presented in Table 5.

Although there have been some studies on the performance analysis and optimization of energy-saving equipment for ships [274], there are still the following problems and challenges.

(1) Further efforts are needed to strengthen the CFD-based coupled dynamics analysis and optimization of energy-saving equipment considering the influence of multiple complex factors comprehensively [275,276].
(2) The interactions among the ship, engine, propeller, and energy-saving appendages using dynamics analysis need to be further studied to assess their impact on the maneuverability and overall energy consumption of ships. Additionally, more studies should be carried out to strengthen the coupled dynamics analysis and joint optimization of the ship, propeller, and energy-saving appendages based on the CFD approaches [277].
(3) Currently, there is relatively limited research on the optimization and application effects analysis of energy-saving appendages using dynamics methods. Therefore, it is necessary to conduct more practical experiments and simulations on different types of energy-saving appendages to achieve the goal of improving ship energy efficiency [278].

Table 5
Performance optimization of energy-saving equipment based on CFD approaches.

Types	References	Optimization parameters	Drag reducing effect
Rudder ball thrust fin	[240]	Propulsion efficiency	The propulsion efficiency is increased by 2.28 %
Rectifier conduit	[241]	Resistance characteristics	The resistance is decreased by approximately 2.49 %
PSS	[242]	Propulsion efficiency	The propulsion efficiency can be improved by 4.69 %
Rudder-Ball-Fin	[243]	Hydrodynamic parameter	The propeller efficiency can be increased by 2.63 %
Hull vane	[263]	Total drag coefficient	The total drag coefficient of ship is reduced by 7 %
GRS	[264]	Ship energy-saving effect	The ship energy saving can be improved by 10 %
Pre-duct	[265]	Propulsion performance	The ship propulsion performance is improved by 3 %
Mewis Duct	[266]	Propeller performance	The propeller thrust can be improved by 3 %–5 %
PD-PSF	[267]	Ship energy-saving effect	The energy saving effect is as much as by 4.26 %
PBCF	[268]	Ship energy-saving effect	The ship propulsion energy can be saved by 2 %

7. Comprehensive performance optimization through CFD-based coupled dynamics analysis

The comprehensive performance optimization through the CFD-based coupled dynamics analysis aims to achieve integrated optimization design [279], and energy efficiency enhancement of hull shapes, propellers, energy-saving devices, and assisted propulsion devices by comprehensively considering the effects of multi-phase coupling of air-liquid-solid [280,281]. The integrated optimization design through CFD-based coupled dynamics analysis is an efficient technique and method to enhance the overall energy efficiency of ships [282,283]. Huang et al. [284] developed a bi-directional transient fluid-solid coupling algorithm, and studied the transient fluid-solid coupling characteristics and laws of composite material propellers. On this basis, the hydrodynamic performance in non-uniform wake fields could be accurately predicted. Zhang et al. [285] analyzed the propulsion performance and changes in wake fields of the composite propeller-rudder system (CPRS) by adopting the coupling analysis method of fluid-solid, which is of great importance for further enhancing the operational performance of the propeller-rudder system (PRS). In addition, Zhang et al. [286] analyzed the coupling effects between the water jet propeller and hull based on CFD approach, as shown in Fig. 29. The results showed that the coupling dynamics optimization of the water jet propeller and the hull can effectively improve the propeller propulsion efficiency. The proposed coupling dynamics analysis method can provide a reference for the optimal design and operation optimization of the water jet propulsion system.

The engine-propeller-hull is a complex system, which has strong coupling relationship in the actual navigation [287]. Integrated coupling dynamics analysis of engine-propeller-hull based on CFD approaches can achieve the optimization design of propulsion systems and enhance the overall energy efficiency of ships. Taskar et al. [288] analyzed the effects of wind and waves on the dynamics performance of the engine-propeller coupling system, which could provide an important foundation for ship fuel efficiency optimization. Liu et al. [289] conducted analysis on the hydrodynamic interaction between the hull-engine-propeller coupling system and analyzed the engine operating characteristics during ship turning circle maneuvers by adopting CFD approaches. Additionally, Liu et al. [290] established a coupling dynamics analysis model of propeller-shafting-hull system and analyzed the coupling effect of propeller dynamics on shafting dynamics based on the CFD approaches. The optimized shafting dynamics with consideration of the coupling effect of propeller-shafting-hull was achieved, which can effectively improve the ship energy efficiency. Furthermore, Song et al. [291] conducted a coupling dynamics analysis of the integrated hull-propeller-rudder-stern flap system by using CFD method, as shown in Fig. 30. The study results showed that the propulsion performance improvement by adopting the stern flap can achieve 50 %–70 % of energy saving. Therefore, the integrated ship-propeller-rudder-stern flap calculation based on CFD approach is significant to further enhance ship fuel efficiency.

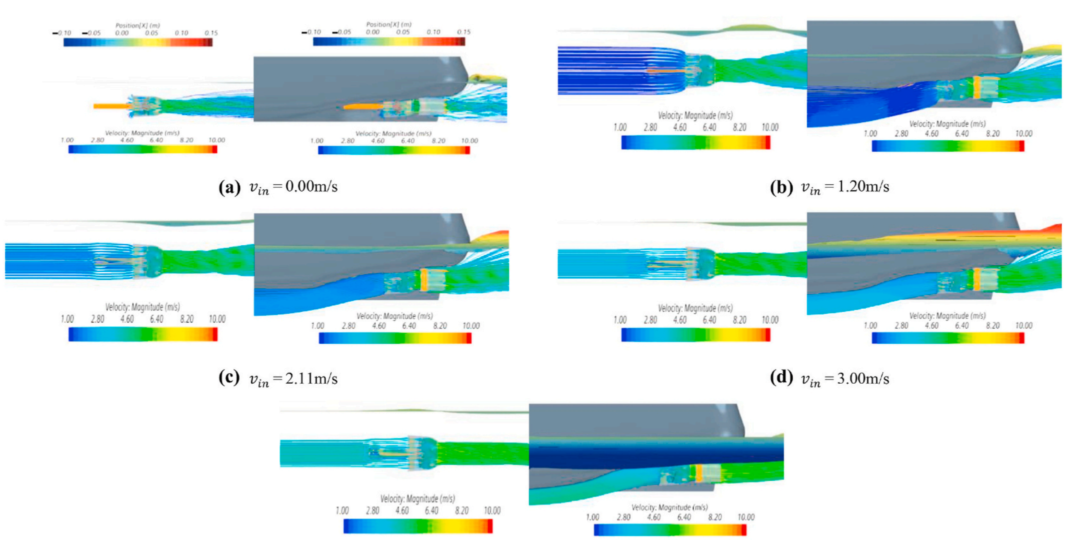
The design optimization of the ship is complex system engineering, because the overall performance and energy efficiency of the ship are closely related to hydrodynamics, aerodynamics, structural mechanics, and other characteristics. The presence of multiphase flow, multiphysics field interactions, and nonlinear effects make the comprehensive optimization design of ships through the CFD-based coupling dynamics analysis extremely complex. Due to limitations in experimental conditions and equipment, it is often challenging to validate the results of multi-field coupling simulation calculations through real experiments. Additionally, the comprehensive optimization design of ships through the CFD-based coupling dynamics analysis struggles to consider multiple parameters, variables, and their highly nonlinear interactions effectively. Therefore, further studies are needed to deepen the dynamics performance analysis and optimization by taking into account the impact of multiple factors and nonlinear coupling effects for the integrated optimization of the ship.

8. Discussions

The CFD-based energy-saving technologies, including the hull optimization design, drag reduction technology, navigation state optimization, efficient propulsion devices, energy-saving equipment, and the coupled dynamics analysis for comprehensive performance optimization have been comprehensively discussed in this research. On this basis, the energy-saving effects of different technologies for the practical applications are summarized, as shown in Table 6.

As can be seen from Table 6, the hull optimization technology can reduce ship resistance by more than 2 %, with the highest of 21.34 % of the total resistance coefficient reduction for the Trimaran. In addition, the hull drag reduction technology can reduce the ship resistance by more than 7 %, and the DR Polymer coating has the maximum drag reduction effect with 82.6 %, meanwhile, increasing the boat sailing speed by 10 %–30 %. Moreover, the navigation state optimization technology can to achieve 10 % of reduction in the ship resistance by optimizing the trim of the KCS. Furthermore, the efficient propulsion devices can effectively improve the ship dynamics performance by more than 2 % and the application of a post-swirl pump jet system can improve the performance of the entire hull by as much as 82 %. Last but not least, the energy-saving equipment can improve the ship propulsion performance by more than 2 %. Particularly, the adoption of the NACA 2415 vane can reduce the total resistance by 20.13 % for a model ship. Overall, the applications of the CFD-based energy-saving technologies can effectively improve the comprehensive performance and energy efficiency level of the ship, thereby contributing to the development of the low-carbon shipping industry.

The energy-saving technologies through CFD-based dynamics analysis can effectively improve ship energy efficiency. In recent years, the



(a) $v_{in} = 0.00$m/s (b) $v_{in} = 1.20$m/s (c) $v_{in} = 2.11$m/s (d) $v_{in} = 3.00$m/s (e) $v_{in} = 3.99$m/s

Fig. 29. Stern wave and streamline between single propeller and propeller-rudder-hull coupling [286].

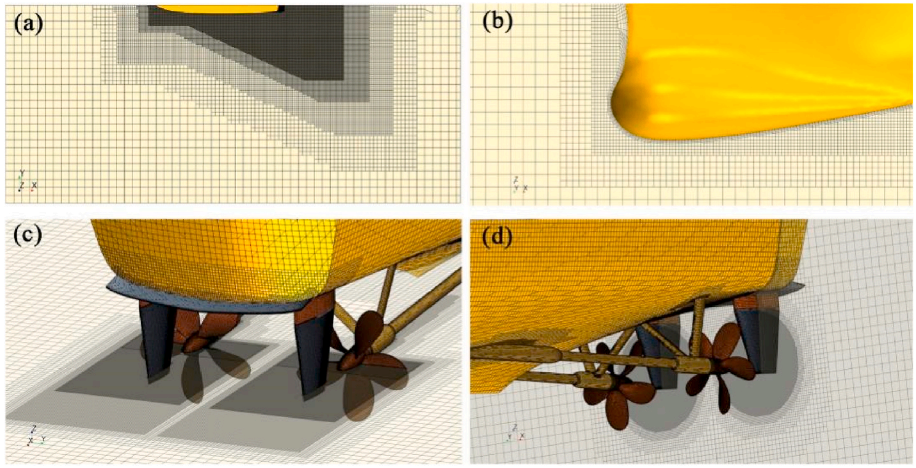


Fig. 30. The integrated coupling dynamics analysis model of the hull-propeller-rudder-stern flap [291].

advancement of AI technology has also promoted the applications of machine learning (ML) in fluid mechanics analysis to achieve the optimal design and energy management of ships [292,293]. Consequently, the ML-based approaches, with the advantages of enhanced accuracy and intelligence as well as high speed calculation [294], have also been adopted for the dynamics analysis to improve the ship energy efficiency [295]. In the optimization of hull shape design, Fahrnholz and Caprace [296] proposed a resistance prediction model based on ML, which can quickly assess hull resistance during the initial design in comparison to the CFD approach. Bagazinski and Ahmed [297] optimized the hull ship based on ML, which can decrease the ship resistance by 60 %. In addition, Zhang et al. [298] proposed a hull shape optimization method based on deep belief network (DBN), and the results showed that the wave-making resistance coefficient can be decreased by 12.6 %, which is closely aligned with the CFD results. Ao et al. [299] proposed an advanced integrated hull optimization method based on artificial intelligence-aided design (AIAD), which can decrease ship resistance by 3 %. As for the optimization of ship propeller design, Li et al. [300] developed a propeller diagram based on CFD and ML, which can improve the propulsion performance by 7 %. Zarezadeh et al. [301] focused on the optimization of surface-piercing propeller (SPP) sections using artificial neural networks (ANN). The results showed that this approach can obtain the optimal SPP section, similarly with the results obtained from CFD, as shown in Fig. 31. In addition, Lee and Lee [302] proposed an ANN-based prediction method for the resistance analysis of the flow control fins (FCFs) with the similar accuracy compared to the CFD results. Additionally, Vasilev et al. [303] proposed a ship trim optimization method based on CFD and ANN methods, which can reduce the daily fuel oil consumption (DFOC) by as much as 10.5 %.

Above all, the ML-based dynamics analysis method can achieve the ship optimal design and energy efficiency improvement [304]. The research on the dynamics analysis based on ML for ship energy saving are summarized, as illustrated in Table 7. As it can be seen, the ML-based dynamics analysis method mainly includes the DBN, ANN, AIAD, DL, which have been applied in hull shape design and propeller optimization with good performance in reducing ship resistance and energy consumption. In addition, the ML-based method can achieve the same effect as the CFD approach to predict the ship performance, meanwhile reducing the computation time, improving the prediction accuracy, and speeding up the convergence of CFD calculation [305,306].

The specific contents of the review mainly include the hull optimization design, drag reduction technology, navigation state optimization, efficient propulsion devices, energy-saving equipment, and the coupled dynamics analysis for comprehensive performance optimization. Although the new advanced energy-saving technologies based on dynamics analysis have been reviewed, there are other CFD-based energy-saving technologies, such as performance optimization of diesel engines. Due to the extensive scope and complexity of this topic, it is not feasible to comprehensively discuss all the energy-saving technologies in this paper. A specific review of those other energy-saving technologies would be carried in the future research work.

9. Conclusions and prospects

The research and applications of energy-saving techniques based on CFD approaches can effectively analyze and optimize the performance of ships in terms of energy efficiency. A comprehensive overview of the current research on the energy-saving techniques based on CFD approaches, including the hull optimization design, drag reduction technology, navigation state optimization, efficient propulsion devices, energy-saving equipment, and the CFD-based coupled dynamics analysis for comprehensive performance optimization is carried out. In addition, the challenges faced in the development of ship fuel efficiency improvement techniques based on CFD analysis methods are comprehensively analyzed, and the future research works on the CFD-based energy efficiency optimization are proposed.

(1) The development and applications of the CFD approaches effectively contribute to the ship design optimization and energy efficiency improvement. However, the problems and challenges including mesh generation difficulties, uncertainty in physical models, appropriate boundary condition selection, significant computational resources, time requirements, and the need for experimental validation to ensure the accuracy and reliability, should be further studied.
(2) The design optimization and energy efficiency improvement of ships based on dynamics analysis methods requires the comprehensive consideration of influencing factors, such as complex marine environments (wind, waves, and currents). Those influencing factors, which are usually difficult to accurately predict and control, have significant impacts on the CFD analysis results. Currently, in the optimization design of ship performance, the coupling impact of complex navigation conditions, load conditions, speed, waves, and ship motion on the comprehensive energy efficiency performance of ships are not fully taken into consideration. Thus, going forward, it is essential to conduct the integrated numerical analysis and optimization design of ships considering the influence of multiple factors and the coupling effects, thereby further reducing ship energy usage and environmental footprint.
(3) The complex and variable influencing factors would result in a certain degree of uncertainty for the CFD analysis and

Table 6
The CFD-based ship energy-saving technologies.

Technology	Method	Research Target	Energy-saving effect	Reference
Hull optimization design	CFD-IME	S60 Hull	The total resistance of the hull can be reduced by 5.15 %	[42]
	CFD-SBO	KCS	The hull shape optimization efficiency can be improved by at least 17.5 %	[43]
	CFD	Ship model	The hull resistance can be decreased by 3 %	[44]
	CFD-MIGA	Trimaran	The total resistance coefficient of the ship can be reduced by 21.34 %	[52]
	CFD-DOE	Fishing ship	The resistance coefficient can be reduced by 2 %	[66]
	CFD	Container ship	The hull resistance can be reduced by 9 %	[71]
	CFD	Japan Bulk Carrier	The resistance can be decreased by 9.48 %	[72]
	CFD-CAD	Three-body ship	The hull resistance can be reduced by 13.3 %	[74]
Hull drag reduction technology	CFD	DR Polymer coating	The maximum drag reduction effect can reach by 82.6 %	[89]
	CFD	Hydrofoil	The resistance can be decreased by 10 %	[95]
	CFD	A river-sea bulk cargo	The total drag coefficient can be reduced by 19 %	[96]
	CFD	A Catamaran ROPAX ferry	The hull resistance can be reduced by 10.45 %	[98]
	CFD	Experimental ship model	The maximum drag reduction effect can reach by 7 %	[106]
	CFD	KCS	The maximum friction resistance of the ship can be reduced by 27.6 %	[112]
	CFD	A ship with an air chamber	Wave resistance can be reduced by 14 %	[115]
	CFD	Experimental ship model	The drag reduction rate is about 32.78 %	[116]
	CFD	High-speed planning boat	The sailing speed can be increased by 10 %–30 %	[117]
Navigation state optimization	CFD	VLCC ship	The total resistance of the ship is the lowest at a bow trim of 0.2°	[154]
	CFD	KCS	The total resistance of the ship is the lowest at a bow trim of 0.6°	[156]
	CFD-BEM	VLCC ship	The trim optimization can achieve fuel saving of 949.3 kg for a voyage	[157]
	CFD	KCS	The resistance can be reduced by 10 % by the trim optimization	[162]
	CFD-DP	A 7500-ton inland bulk carrier	The total ship power consumption can be reduced by 7.64 %	[167]
Efficient propulsion devices	CFD	The propeller model of JBC	The propeller wake performance can be improved by 2 %	[178]
	CFD	A new innovative backflow marine propeller	The efficiency of the optimized propeller can be improved by 8 %–9 %	[179]
	CFD	Wageningen B-series propeller	The ship efficiency can be improved by 62 % at lower propeller thrust and torque coefficients	[181]
	CFD	The rim-driven thruster	The optimized rim-driven thruster propulsion efficiency can be improved by 3.3 %	[210]
	CFD	Ka4-70 propeller	The optimized rim-driven thruster can improve open-water efficiency by 10.9 %	[211]
	CFD	Shaftless rim thruster	The overall efficiency of the thruster can be increased by 5.79 %	[213]
	CFD	A post-swirl pump jet system	The performance of the entire hull can be improved by 82 %	[221]
	CFD	The podded propulsion cruise ship	The maximum propulsion efficiency of the podded propulsion is 0.7010 at the initial installation position	[228]
Energy-saving equipment	CFD	A single screw 35000 DWT bulk carrier	The model-scale simulation showed a 4.85 % improvement in ship propulsion efficiency	[240]
	CFD	Japan bulk carrier	The ship resistance can be reduced by 2.49 %	[241]
	CFD-FEM	KVLCC2 ship	The PSS can improve the efficiency of the propellers by 4.69 %	[242]
	CFD-EFD	A single screw 35000 DWT bulk carrier	The rudder-bulb-fins system can improve the ship propulsion efficiency by 2.63 %	[243]
	CFD	Ship model equipped with NACA 2415 vane	The total resistance of the ship can be reduced by 20.135 % compared with that of the ship without hull vane	[262]
	CFD	A high-speed displacement vessel model	The total drags coefficient of the ship equipped with hull vane can be reduced by up to 7 %	[263]
	CFD	6400 DWT general cargo ship	The GRS can improve the ship energy saving by 10 %	[264]
	CFD	Fishing boat	The use of Pre-duct can improve the ship propulsion performance by 3 %	[265]
	CFD	INSEAN e779a propeller	The use of Mewis Duct can improve the propeller thrust by 3 %–5 %	[266]
	CFD	A single screw 35000DWT bulk carrier	The energy-saving effect of the ship is as much as by 4.26 %	[267]
	CFD	Wageningen B series propeller	The PBCF can improve the energy saving of the propulsion system by 2 %	[268]
	CFD-PSO	New Aden	The system thrust coefficient can be increased by 6.5 %	[269]

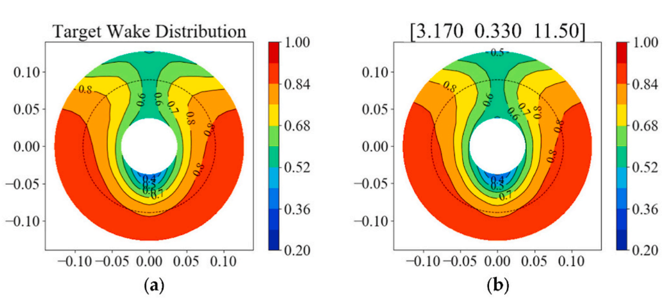


Fig. 31. Comparison of ANN and CFD for wake distributions for ship optimal design [301].

calculation, thereby affecting the effectiveness and accuracy of the model and analysis. Therefore, it is essential to consider the synergy between different ship systems and components, as well as the diversity of ship structures in the dynamics performance analysis. It is essential to establish a comprehensive ship operation performance analysis method considering complex and uncertain dynamic factors. It is also vital to analyze the impact of irregular waves on the overall performance of ships and strengthen the research on high Reynolds number turbulence models. In addition, to solve the problem of model analysis uncertainty, more accurate boundary conditions and reasonable mesh division methods should be employed.

(4) The integrated optimization design of the ship based on the CFD-based coupled dynamics analysis is an effective way and method to improve ship fuel efficiency. In the future, it is necessary to strengthen the fundamental research on multi-field coupled dynamics, conduct numerical simulation calculations on multiple typical ships and multi-flow field conditions, analyze the coupled dynamic characteristics of the integrated ship-engine-propeller-appendages system, and establish a gas-liquid-solid multiphase coupled dynamics analysis method. It will provide effective

Table 7
The dynamics analysis based on ML for ship energy saving.

Method	Research Target	Effect	Reference
ML	KCS	The resistance of the ship can be reduced by 60 %	[298]
DBN	Wigley ship	The wave-making resistance coefficient can be reduced by 12.6 %	[299]
AIAD	KCS	The resistance of the ship can be decreased by 3 %	[300]
ML	INSEAN E1619 propeller	The propulsion performance of the propeller can be improved by 7 %	[301]
ANN	Surface piercing propeller	The relative errors of ANN and CFD results in propeller thrust and torque are 3.5 % and 4.24 %, respectively	[302]
ANN	FCFS on 1000 TEU container ship	The prediction result of viscous resistance coefficient is less than 0.01 % of the CFD results	[303]
ANN	Ro-Ro vessel	The DFOC can be reduced by as much as 10.5 % at a 1.5m trim and 7.5m draft	[304]
DL	P5475 propeller	The cavitation volume of the propeller can be reduced as much as by 51 %	[307]

technical solutions for enhancing the ship fuel efficiency by adopting the CFD-based coupled dynamics analysis approaches.

(5) Currently, ship performance optimization research is mainly based on numerical simulation analysis using dynamics theory, and there are lacks of effective testing and verification analysis, especially for new energy-saving technologies and devices. Thus, going forward, it is essential to strengthen the experimental study on ship fuel efficiency enhancement based on CFD approaches, propose the optimization performance test and energy-saving evaluation methods for the CFD-based ship optimization design, establish a software system for the ship performance optimization design and energy-saving evaluation, and develop more advanced numerical methods and computing technologies. It will improve the consistency between laboratory simulation experiments and actual sailing environment, and thus ensuring the reliability and authenticity of laboratory numerical simulation calculations.

(6) In the ship optimal design and performance optimization, there is still lack of analysis on the interaction between the ship-engine-propeller-appendages and their impact on the overall operational performance in terms of ship fuel efficiency. Therefore, it is necessary to carry out the CFD-based coupled dynamics analysis of the ship-engine-propeller-appendages under the cross-coupling effects of multiphase flow and multi-physics field, and propose an energy efficiency improvement method for integrated optimization design of the ship, and develop an integrated optimization system for the ship-engine-propeller-appendages design based on the CFD-based coupled dynamics analysis, thus improving the overall fuel efficiency of the ship.

Declaration of competing interest

The authors declare that they have no known competing financial interests or personal relationships that could have appeared to influence the work reported in this paper.

Data availability

The data that has been used is confidential.

Acknowledgements

The authors are grateful to the support of the National Key R&D Program of China (2022YFB4300803), the National Natural Science Foundation of China (52271305, 52071045), and the Fundamental Research Funds for the Central Universities (3132023525, 3132024207).

References

[1] Hussain MD, Amin OM. A comprehensive analysis of the stability and powering performances of a hard sail–assisted bulk carrier. J Mar Sci Appl 2021;20:426–45.
[2] Hoang TA, Pandey A, Martinez De Osés FJ, Chen WH, Said Z, et al. Technological solutions for boosting hydrogen role in decarbonization strategies and net-zero goals of world shipping: challenges and perspectives. Renew Sustain Energy Rev 2023;188:113790.
[3] Mondejar ME, Andreasen JG, Pierobon L, Larsen U, Thern M, Haglind F. A review of the use of organic Rankine cycle power systems for maritime applications. Renew Sustain Energy Rev 2018;91:126–51.
[4] Wang K, Li J, Yan X, Huang L, Jiang X, Yuan Y, et al. A novel bi-level distributed dynamic optimization method of ship fleets energy consumption. Ocean Eng 2020;197:106802.
[5] Bilgili L. Comparative assessment of alternative marine fuels in life cycle perspective. Renew Sustain Energy Rev 2021;144:110985.
[6] Bilgili L. A systematic review on the acceptance of alternative marine fuels. Renew Sustain Energy Rev 2023;182:113367.
[7] Yuan Y, Wang J, Yan X, Shen B, Long T. A review of multi-energy hybrid power system for ships. Renew Sustain Energy Rev 2020;132:110081.
[8] Wang K, Hua Y, Huang L, Guo X, Liu X, Ma Z, Ma R, Jiang X. A novel GA-LSTM-based prediction method of ship energy usage based on the characteristics analysis of operational data. Energy 2023;282:128910.
[9] Liu S, Song J, Zhang L. Research on 3-DOF stabilized control system of ship propulsion-assisting sail. Ocean Eng 2023;278:114370.
[10] Korberg AD, Brynolf S, Grahn M, Skov IR. Techno-economic assessment of advanced fuels and propulsion systems in future fossil-free ships. Renew Sustain Energy Rev 2021;142:110861.
[11] Fan A, Yan J, Xiong Y, Shu Y, Fan X, Wang Y, He Y, Chen J. Characteristics of real-world ship energy consumption and emissions based on onboard testing. Mar Pollut Bull 2023;194(A):115411.
[12] Inal OB, Charpentier JF, Deniz C. Hybrid power and propulsion systems for ships: current status and future challenges. Renew Sustain Energy Rev 2022;156:111965.
[13] Wang H, Zhu R, Gu M, Gu X. Numerical investigation on steady wave of high-speed ship with transom stern by potential flow and CFD methods. Ocean Eng 2022;247:110714.
[14] Li S, Liu C, Chu X, Zheng M, Wang Z, Kan J. Ship maneuverability modeling and numerical prediction using CFD with body force propeller. Ocean Eng 2022;264:112454.
[15] Kim D, Yim J, Song S, Demirel YK, Tezdogan T. A systematic investigation on the manoeuvring performance of a ship performing low-speed manoeuvres in adverse weather conditions using CFD. Ocean Eng 2022;263:112364.
[16] Li F, Yuan Y, Yan X, Malekian R, Li Z. A study on a numerical simulation of the leakage and diffusion of hydrogen in a fuel cell ship. Renew Sustain Energy Rev 2018;97:177–85.
[17] Kim D, Tezdogan T. CFD-based hydrodynamic analyses of ship course keeping control and turning performance in irregular waves. Ocean Eng 2022;248:110808.
[18] Windt C, Davidson J, Ringwood JV. High-fidelity numerical modelling of ocean wave energy systems: a review of computational fluid dynamics-based numerical wave tanks. Renew Sustain Energy Rev 2018;93:610–30.
[19] Thai GT, Quang VH, Hyun CK. Optimization strategy for planning hull design. Int J Nav Archit Ocean Eng 2022;14:100471.
[20] Zhong K, Ni B, Li Z, Xu X, Wei H, Li H, Xue Y. Direct measurements and CFD simulations on ice-induced hull pressure of a ship in floe ice fields. Ocean Eng 2023;272:113523.
[21] Peri D, Rossetti M, Campana EF. Design optimization of ship hulls via CFD techniques. J Ship Res 2001;45(2):140–9.
[22] Majidian H, Ahani E, Wang L. Aerodynamic study of pyramid stacking configuration on mega-sized container ships using CFD to optimize lateral drag. Appl Ocean Res 2022;124:103209.
[23] Kaidi S, Smaoui H, Sergent P. Numerical estimation of bank-propeller-hull interaction effect on ship manoeuvring using CFD method. Journal of Hydrodynamics, Ser. B 2017;29(1):154–67.
[24] Andersson J, Shiri AA, Rickard Bensow E, Jin Y, Wu Q, Geng Y, Deng G, et al. Ship-scale CFD benchmark study of a pre-swirl duct on KVLCC2. Appl Ocean Res 2022;123:103134.
[25] Saha GK, Miazee MA. Numerical and experimental study of resistance. Sinkage and trim of a container ship. Procedia Eng 2017;194:67–73.
[26] Farkas A, Degiuli N, Martić I. An investigation into the effect of hard fouling on the ship resistance using CFD. Appl Ocean Res 2020;100:102205.
[27] Kim K, Choi S, Seo J, Na KI, Shin HR, Kim DH. Validation of 4DOF maneuvering coefficients optimization using hydrodynamic force and moment estimated from free-running model test results. Int J Nav Archit Ocean Eng 2024;16:100599.
[28] Rahaman MM, Islam H, Akimoto H, Islam MR. Motion predictions of ships in actual operating conditions using potential flow based solver. J Nav Architect Mar Eng 2017;14(1):65–76.
[29] Saydam AZ, Taylan M. Evaluation of wind loads on ships by CFD analysis. Ocean Eng 2018;158:54–63.

[30] Chang H, Liu Z, Zhan C, Feng B, Cheng X, Zheng Q. Sampling method for hull form optimization based on the morphing method and its application. Ocean Eng 2023;281:114715.
[31] Grigoropoulos GJ, Bakirtzoglou C, Papadakis G, Ntouras D. Mixed-Fidelity design optimization of hull form using CFD and potential flow solvers. J Mar Sci Eng 2021;9(11):1234.
[32] Oyuela S, Ojeda HRD, Arribas FP, Otero AD, Sosa R. Investigating fishing vessel hydrodynamics by using EFD and CFD tools, with focus on total ship resistance and its components. J Mar Sci Eng 2024;12(4):622.
[33] Wang S, Duan F, Li Y, Xia Y, Li Z. An improved radial basis function for marine vehicle hull form representation and optimization. Ocean Eng 2022;260:112000.
[34] Zhao Y, Zong Z, Zou L. Ship hull optimization based on wave resistance using wavelet method. Journal of Hydrodynamics, Ser. B 2015;27(2):216–22.
[35] Sulistyawati W, Suranto JP, Purnamasari D. Hull-shape parametric for minimising wave energy. Energy Rep 2023;9(1):487–94.
[36] Pranesh SB, Kumar D, Anantha Subramanian V, Sathianarayanan D, Ramadass GA. Non-linear buckling analysis of imperfect thin spherical pressure hull for manned submersible. J Ocean Eng Sci 2017;2(4):293–300.
[37] Yang C, Huang F. An overview of simulation-based hydrodynamic design of ship hull forms. Journal of Hydrodynamics, Ser. B 2016;28(6):947–60.
[38] Brizzolara S, Curtin T, Bovio M, Vernengo G. Concept design and hydrodynamic optimization of an innovative SWATH USV by CFD methods. Ocean Dynam 2012; 62(2):227–37.
[39] Wang C, Yang M, Wang Y, Ren M, Wang Z, Yang S. Design and optimization of cylindrical hull with non-uniform arch ribs for underwater gliders based on approximate model and experiments. Ocean Eng 2022;259:111831.
[40] Zha L, Zhu R, Hong L, Huang S. Hull form optimization for reduced calm-water resistance and improved vertical motion performance in irregular head waves. Ocean Eng 2021;233:109208.
[41] Kim H, Yang C. A new surface modification approach for CFD-based hull form optimization. Journal of Hydrodynamics, Ser. B 2010;22(5):520–5.
[42] Ouyang X, Chang H, Feng B, Liu Z, Zhan C, Cheng X. Application of an improved maximum entropy sampling method in hull form optimization. Ocean Eng 2023; 270:112702.
[43] Wei YB, Chen X, Wang JH, Wan DC. Multi-objective hull form optimization utilizing sequential sampling optimization method. Ocean Eng 2024;310(1): 118667.
[44] Cheng XK, Wang D, Huang XH, Yan X, Dou P, Ji RW. Comparative investigations on hull line optimization based on traditional and fully parametric methods for saving energy. Front Energy Res 2024;12:1359957.
[45] Nazemian A, Ghadimi P. CFD-based optimization of a displacement trimaran hull for improving its calm water and wavy condition resistance. Appl Ocean Res 2021;113:102729.
[46] Nazemian A, Ghadimi P. Shape optimisation of trimaran ship hull using CFD-based simulation and adjoint solver. Ships Offshore Struct 2020;17(2):359–73.
[47] Senov AA, Pechenyuk AV, Vučinić D. Ship hull form design and optimization based on cfd. Towards green marine technology and transport - proceedings of the 16th international congress of the international maritime association of the mediterranean. IMAM; 2015. p. 215–23.
[48] Siddiqui MA, Greco M, Lugni C, Faltinsen OM. Experimental studies of a damaged ship section in beam sea waves. Appl Ocean Res 2020;97:102090.
[49] Jang CD, Seo SI, Kim SK. A study on the optimum structural design of surface effect ships. Mar Struct 1996;9(5):519–44.
[50] Song S, Kim D, Dai S. CFD investigation into the effect of GM variations on ship manoeuvring characteristics. Ocean Eng 2024;291:116472.
[51] Gadelho JFM, Rodrigues JM, Lavrov A, Soares CG. Heave and sway hydrodynamic coefficients of ship hull sections in deep and shallow water using Navier-Stokes equations. Ocean Eng 2018;154:262–76.
[52] Zong Z, Hong ZC, Wang YG, Hefazi H. Hull form optimization of trimaran using self-blending method. Appl Ocean Res 2018;80:240–7.
[53] Miao A, Zhao M, Wan D. CFD-based multi-objective optimisation of S60 Catamaran considering Demihull shape and separation. Appl Ocean Res 2020;97: 102071.
[54] Cheng X, Feng B, Liu Z, Chang H. Hull surface modification for ship resistance performance optimization based on Delaunay triangulation. Ocean Eng 2018; 153:333–444.
[55] Zhang Y, Li X, Ma S, Shao W, Zhang Y. Numerical investigation on the water entry of bow-flare section with different bulbous bow shapes. Ocean Eng 2022;257: 111715.
[56] Samuel S, Wicaksono A, Kurniawan WA, Hadi ES, Tuswan T, Trimulyono A, Muryadin M. Investigation of an inverted bow on frigate hull resistance. J Appl Fluid Mech 2023;17(1):136–47.
[57] Silva-Campillo A, Ulla-Campos L, Suárez-Bermejo JC, Herreros-Sierra MA. Effect of bow hull form on the buckling strength assessment of the corner bracket connection. Ocean Eng 2022;265:112562.
[58] Tran TG, Huynh QV, Kim HC. Optimization strategy for planing hull design. Int J Nav Archit Ocean Eng 2022;14:100471.
[59] Lee CM, Yu JW, Choi JE, Lee I. Effect of bow hull forms on the resistance performance in calm water and waves for 66k DWT bulk carrier. Int J Nav Archit Ocean Eng 2019;11(2):723–35.
[60] Liu X, Zhao W, Wan D. Hull form optimization based on calm-water wave drag with or without generating bulbous bow. Appl Ocean Res 2021;116:102861.
[61] Yu JW, Lee CM, Lee I, Choi JE. Bow hull-form optimization in waves of a 66,000 DWT bulk carrier. Int J Nav Archit Ocean Eng 2017;9(5):499–508.
[62] Luo W, Lan L. Design optimization of the lines of the bulbous bow of a hull based on parametric modeling and computational fluid dynamics calculation. Math Comput Appl 2017;22(1):4.
[63] Grigoropoulos GJ. Hydrodynamic design of ships. J Mar Sci Eng 2022;10(4):512.
[64] Yuan WX, Wall A, Thornhill E, Sideroff C, Mamou M, Lee R. CFD aided ship design and helicopter operation. J Mar Sci Eng 2022;10(9):1304.
[65] Cheng X, Feng B, Chang H, et al. Multi-objective optimisation of ship resistance performance based on CFD. J Mar Sci Technol 2019;24:152–65.
[66] Hong Z, Zong Z, Li H, Hefazi H, Sahoo PK. Self-blending method for hull form modification and optimization. Ocean Eng 2017;146:59–69.
[67] Asmara IPS, Sumardiono Adam M. Seakeeping and resistance analysis of 1200GT passenger ship fitted with NACA 4412 stern foil using CFD method. IOP conference series: materials science and engineering, the international conference on maritime technology and its applications (ICOMTA 2020) 12th december 2020, surabaya, vol. 1175; 2020, 012002. Indonesia.
[68] Wang X, Liu L, Zhang Z, Feng D. Numerical study of the stern flap effect on catamaran' seakeeping characteristic in regular head waves. Ocean Eng 2020; 206:107172.
[69] Wu P, Chen J, Liu X, Wu C, Lu C. Sonar dome geometry design using CFD to reduce ship resistance at cruise speed. Sensors 2022;22(14):5342.
[70] Chen X, Diez M, Kandasamy M, Zhang Z, Campana EF, Stern F. High-fidelity global optimization of shape design by dimensionality reduction, metamodels and deterministic particle swarm. Eng Optim 2015;47(4):473–94.
[71] Chen J, Wei J, Jiang W. Optimization of a twin-skeg container vessel by parametric design and CFD simulations. Int J Nav Archit Ocean Eng 2016;8(5): 466–74.
[72] Liu Z, Zhao W, Wan D. Resistance and wake distortion optimization of JBC considering ship-propeller interaction. Ocean Eng 2022;244:110376.
[73] Lu Y, Chang X, Yin X. Hydrodynamic design study on ship bow and stern hull form synchronous optimization covering whole speeds range. Math Probl Eng 2019;2019:1–19.
[74] Hamed A. Multi-objective optimization method of trimaran hull form for resistance reduction and propeller intake flow improvement. Ocean Eng 2022; 244:110352.
[75] Chuah LF, Mokhtar K, Mhd Ruslan SM, Abu Bakar A, Abdullah MA, Osman NH, et al. Implementation of the energy efficiency existing ship index and carbon intensity indicator on domestic ship for marine environmental protection. Environ Res 2023;222:115348.
[76] Orlandi A, Calastrini F, Kalikatzarakis M, Guarnieri F, Busillo C, Coraddu A. Air quality forecasting of along-route ship emissions in realistic meteo-marine scenarios. Ocean Eng 2024;291:116464.
[77] Marlantes KE, Maki KJ. A hybrid data-driven model of ship roll. Ocean Eng 2024; 303:117821.
[78] Kim YC, Kim KS, Yeon S, Lee YY, Kim GD, Kim M. Power prediction method for ships using data regression models. J Mar Sci Eng 2023;11(10):1961.
[79] Lee S, You JM, Lee HH, Lim T, Rhee SH, Rhee KP. Preliminary tests of a damaged ship for CFD validation. Int J Nav Archit Ocean Eng 2012;4(2):172–81.
[80] Campos RM, Islam H, Ferreira TRS, Soares CG. Impact of heavy biofouling on a nearshore heave-pitch-roll wave buoy performance. Appl Ocean Res 2021;107: 102500.
[81] Reis Oliveira D, Granhag L. Ship hull in-water cleaning and its effects on fouling-control coatings. Biofouling 2020;36(3):332–50.
[82] Rowin WA, Asha AB, Narain R, Ghaemi S. A novel approach for drag reduction using polymer coating. Ocean Eng 2021;240:109895.
[83] Nugroho FA, Garhuom W, Andresen-Paulsen G, Ehlers S. Numerical analysis of the correlation between the pitting severity and surface roughness of corroded specimens. Ships Offshore Struct 2022;171(2):2699–714.
[84] Owen D, Demirel YK, Oguz E, Tezdogan T, Incecik A. Investigating the effect of biofouling on propeller characteristics using CFD. Ocean Eng 2018;159:505–16.
[85] Yeginbayeva IA, Granhag L, Chernoray V. Review and historical overview of experimental facilities used in hull coating hydrodynamic tests. Proc IME M J Eng Marit Environ 2019;233(4):1240–59.
[86] Kim BS, Kim Y. CFD application for prediction of ship added resistance in waves. 13th ISOPE Pacific/Asia offshore mechanics symposium, PACOMS. 2018. p. 9–15.
[87] Yusuf Y, Ghazali MJ, Taha MM, Omar NI. A brief overview of the marine environmentally friendly anti-fouling surface strategy. Proc IME J J Eng Tribol 2023;237(6):1384–93.
[88] Zimbelmann S, Emde B, Heusinger von Waldegge T, Stübing D, Baumann M, Hermsdorf J. Interaction between laser radiation and biofouling for ship hull cleaning. Procedia CIRP 2022;111:705–10.
[89] Du XR, Mutsuda H, Wasada Y, Nakashima T. Hybrid simulation of dissipative particle dynamics and computational fluid dynamics for friction drag reduction of polymer coatings. Ocean Eng 2023;285:115415.
[90] Kim HN, Jeong SW, Baik BG, Jang H, Jeong HE, Lee SJ. Bioinspired low-friction surface coating with lubricant-infused spherical cavities for sustainable drag reduction. Appl Surf Sci 2023;628:157365.
[91] García S, Trueba A, Boullosa-Falces D, Islam H, Soares CG. Predicting ship frictional resistance due to biofouling using Reynolds-averaged Navier-Stokes simulations. Appl Ocean Res 2020;101:102203.
[92] Alza Izaguirre P, Rojas LP, Núñez Basáñez JF. Drag reduction trough special paints coated on the hull. International conference on ship drag reduction. 2010.
[93] Ceccio SL. Friction drag reduction of external flows with bubble and gas injection. Annu Rev Fluid Mech 2010;42:183–203.
[94] Kim YR, Steen S. Potential energy savings of air lubrication technology on merchant ships. Int J Nav Archit Ocean Eng 2023;15:100530.

[95] Yanuar Y, Putra MSG, Akbar M, Alief M, Fatimatuzzahra F. Numerical study on influence of hydrofoil clearance towards total drag reduction on winged air induction Pipe for air lubrication. International Journal of Technology 2020;11 (1):91–9.
[96] Yang F, Wang L, Wang J, Chen S, Luo L, Wang M. Numerical simulation on drag reduction of river-sea bulk cargo by gas film. Proceedings of the international offshore and polar engineering conference. 2017. p. 839–43.
[97] Yanuar Waskito KT, Pratama SY, Candra BD, Rahmat BA. Comparison of microbubble and air layer injection with porous media for drag reduction on a self-propelled barge ship model. J Mar Sci Appl 2018;17(2):165–72.
[98] Mohammadpour J, Salehi F, Garaniya V, et al. Computational analysis of air bubble-induced frictional drag reduction on ship hulls. J Mar Sci Technol 2024. https://doi.org/10.1007/s00773-024-01016-0.
[99] Pang M, Zhang Z. Numerical investigation on turbulence drag reduction by small bubbles in horizontal channel with mixture model combined with population balance model. Ocean Eng 2018;162:80–97.
[100] Verschoof RA, Van Der Veen RC, Sun C, Lohse D. Bubble drag reduction requires large bubbles. Phys Rev Lett 2016;117:104502.
[101] Wang H, Wang K, Liu G. Drag reduction by gas lubrication with bubbles. Ocean Eng 2022;258:111833.
[102] Giernalczyk M, Kaminski P. Assessment of the propulsion system operation of the ships equipped with the air lubrication. System Sensors 2021;21(4):1357.
[103] Moriguchi Y, Kato H. Influence of microbubble diameter and distribution on frictional resistance reduction. Marine Science and Technology 2002;7(2):79–85.
[104] Zhao X, Zong Z, Jiang Y, Sun T. A numerical investigation of the mechanism of air-injection drag reduction. Appl Ocean Res 2020;94:101978.
[105] Murai YC, Sakamaki H, Kumagai I, Park HJ, Tasaka Y. Mechanism and performance of a hydrofoil bubble generator utilized for bubbly drag reduction ships. Ocean Eng 2020;216:108085.
[106] Zhao X, Zong Z. Experimental and numerical studies on the air-injection drag reduction of the ship model. Ocean Eng 2022;251:111032.
[107] Zhao XJ, Zong Z, Jiang YC, et al. Numerical simulation of micro-bubble drag reduction of an axisymmetric body using OpenFOAM. J Hydrodyn 2019;31: 900–10.
[108] Tanaka T, Oishi Y, Park HJ, Tasaka Y, Murai Y, Kawakita C. Repetitive bubble injection promoting frictional drag reduction in high-speed horizontal turbulent channel flows. Ocean Eng 2021;239:109909.
[109] Mäkiharju SA, Ceccio SL. On multi-point gas injection to form an air layer for frictional drag reduction. Ocean Eng 2018;147:206–14.
[110] Zhang J, Yang S, Liu J. Numerical investigation of frictional drag reduction with an air layer concept on the hull of a ship. J Hydrodyn 2020;32:591–604.
[111] Mäkiharju SA, Perlin M, Ceccio SL. On the energy economics of air lubrication drag reduction. Int J Nav Archit Ocean Eng 2012;4(4):412–22.
[112] Gamal M, Kotb M, Naguib A, et al. Numerical investigations of micro bubble drag reduction effect for container ships. Marine System & Ocean Technology 2021;16: 199–212.
[113] Matveev KI, Perry NI, Mattson AW, Chaney CS. Development of a remotely controlled testing platform with low-drag air-ventilated hull. J Mar Sci Appl 2015;14(1):25–9.
[114] Cucinotta F, Guglielmino E, Sfravara F. Numerical and experimental investigation of a planning air cavity ship and its air layer evolution. Ocean Eng 2018;152: 130–44.
[115] Choi JK, Hsiao CT, Chahine G. Design trade-off analysis for high performance ship hull with air plenums. 8th international conference on numerical ship hydrodynamics. 2005.
[116] Wu H, Dong WC, Ou YP. Numerical method investigation of drag reduction with air layer at bottom of ship. J Nav Univ Eng 2016;28(3):70–5 (in Chinese).
[117] Matveev KI. Numerical study of shallow-draft air-cavity hull with two-step bottom in deep and shallow water. Ocean Eng 2022;246:110621.
[118] Hasegawa M, Sakaue H. Microfiber coating for drag reduction by flocking technology. Coatings 2018;8(12):464.
[119] Gao QD, Lu JS, Zhang GL, Zhang JW, Wu WF, Deng JJ. Experimental study on bubble drag reduction by the turbulence suppression in bubble flow. Ocean Eng 2023;272:113804.
[120] Nguyen HT, Lee SW, Ryu J, et al. Numerical investigation of the effect of air layer on drag reduction in channel flow over a superhydrophobic surface. Science Report 2024;14:12053.
[121] Jung T, Choi H, Kim J. Effects of the air layer of an idealized superhydrophobic surface on the slip length and skin-friction drag. J Fluid Mech 2016;790:R1.
[122] Yang JW, Park H, Chun HH, Ceccio SL, Perlin M, Lee I. Development and performance at high Reynolds number of a skin-friction reducing marine paint using polymer additives. Ocean Eng 2014;84:183–93.
[123] Busch J, Barthlott W, Brede M, Terlau W, Mail M. Bionics and green technology in maritime shipping: an assessment of the effect of Salvinia air-layer hull coatings for drag and fuel reduction. Phil Trans Math Phys Eng Sci 2019;377:2138.
[124] Cho Y, Jeon KH, Lee SB, et al. Evaluation of in-service speed performance improvement by means of FDR-AF (frictional drag reducing anti-fouling) marine coating based on ISO19030 standard. Science Report 2021;11:1062.
[125] Lee I, Park H. Prediction and verification of full-scale ship performance of frictional drag reduction coating. 10th international symposium on turbulence and shear flow phenomena. 2017.
[126] Kodama Y, Hinatsu M, Hori T, Kawashima H, et al. A full-scale air lubrication experiment using a large cement carrier for energy saving. In: Ohta conference proceedings the Japan society of naval architects and ocean engineers. vol. 6; 2008. p. 163–6.
[127] Zhu DY, SonYX Gao FQ, Dong SS, Xu CH, Liu BH, Zheng JW, Zhou XQ, Liu Q. Achieving underwater stable drag reduction on superhydrophobic porous steel via active injection of small amounts of air. Ocean Eng 2024;308:118329.
[128] Kumagai I, Takahashi Y, Murai Y. Power-saving device for air bubble generation using a hydrofoil to reduce ship drag: theory, experiments, and application to ships. Ocean Eng 2015;95:183–94.
[129] Butterworth J, Atlar M, Shi WC. Experimental analysis of an air cavity concept applied on a ship hull to improve the hull resistance. Ocean Eng 2015;110(B): 2–10.
[130] Park SH, Lee I. Optimization of drag reduction effect of air lubrication for a tanker model. Int J Nav Archit Ocean Eng 2018;10(4):427–38.
[131] Sinha T. Various methods used to minimise resistance on ship's hull. Naval Architecture; 2016.
[132] Li M, Xiao WB, Yin ZZ, Chen YH, Luo YD, Hong Z, Xue MS. Construction of a robust MOF-based superhydrophobic composite coating with the excellent performance in antifouling, drag reduction, and organic photodegradation. Prog Org Coating 2024;186:108086.
[133] Cao Z, Cao P. Research progress on low-surface-energy antifouling coatings for ship hulls: a review. Biomimetics 2023;8(6):502.
[134] García-Magariño A, Lopez-Gavilan P, Sor S, Terroba F. Micro/bubble drag reduction focused on new applications. J Mar Sci Eng 2023;11(7):1315.
[135] Shi YZ, Luo K, Chen XP, Li DJ, Jia LB. A new cavitation model considering inter-bubble action. Int J Nav Archit Ocean Eng 2021;13:566–74.
[136] Ye Q, Ou Y, Xiang G, Chen J. Numerical study on the influence of water depth on air layer drag reduction. Appl Sci 2024;14(1):431.
[137] Wu H, Ou YP. Analysis of air layer shape formed by air injection at the bottom of flat plate. Ocean Eng 2020;216:108091.
[138] Zhang B, Wang S, Deng L, Jia M, Xu J. Ship motion attitude prediction model based on IWOA-TCN-Attention. Ocean Eng 2023;272:113911.
[139] Pérez-Canosa JM, Orosa JA, Galdo MIL, Barros JJC. A new theoretical dynamic analysis of ship rolling motion considering navigational parameters, loading conditions and sea state conditions. J Mar Sci Eng 2022;10:1646.
[140] Gourlay T, Lilienthal T. Dynamic stability of ships in waves. In: Proceedings of the pacific 2002 international maritime conferencia; 2022. p. 29–31. Sydney, Australia.
[141] Luo X, Yan R, Wang SA. Comparison of deterministic and ensemble weather forecasts on ship sailing speed optimization. Transport Res Transport Environ 2023;121:103801.
[142] Wu H. Numerical study of the effect of ship attitude on the perform of ship with air injection in bottom cavity. Ocean Eng 2019;186:106119.
[143] Martić I, Anušić B, Degiuli N, Grlj CG. Numerically investigating the effect of trim on the resistance of a container ship in confined and shallow water. Appl Sci 2024;14(15):6570.
[144] Fritz F, Gatin I. Combining log data and CFD in trim tables. Naval Architect; 2023. p. 38–41.
[145] Wei W, Fu SX, Moan T, Song HC, Ren TX. A time-domain method for hydroelasticity of very large floating structures in inhomogeneous sea conditions. Marine structure 2018;57:180–92.
[146] Zhang L, Du CS, Ni YS, Shang YC, Zhang JN. Mechanism of speed loss reduction and propulsion efficiency improvement of ONR tumblehome with active-controlled stern flaps in resonance waves. J Mar Sci Eng 2024;12(5):822.
[147] Martić I, Degiuli N, Farkas A, Gospić I. Evaluation of the effect of container ship characteristics on added resistance in waves. J Mar Sci Eng 2020;8(9):696.
[148] Härting A, Laupichler A, Reinking J. Considerations on the squat of unevenly trimmed ships. Ocean Eng 2009;36(2):193–201.
[149] Bu S, Zhang P, Liu H, Shi Y, Gu M. Researches on the motion and flooding process of a damaged passenger ship in regular and irregular beam waves. Ocean Eng 2023;281:114927.
[150] Himaya AN, Sano M. Course-Keeping performance of a container ship with various draft and trim conditions under wind disturbance. J Mar Sci Eng 2023;11 (5):1052.
[151] Yu YY, Zhang HS, Mu ZB, Li YT, Sun YT, Liu J. Trim and engine power joint optimization of a ship based on minimum energy consumption over a whole voyage. J Mar Sci Eng 2024;12(3):475.
[152] Wang J, Wan D, Abdel-Maksoud M. Cfd investigations of ship propulsion performance at different trim angles. Proceedings of the international offshore and polar engineering conference. 2020. p. 2338–44.
[153] Mahmoodi H, Hajivand A. Numerical trim and draft optimization of a twin-screw modern surface combatant with inverted bow. Appl Ocean Res 2022;123:103186.
[154] Korkmaz KB, Werner S, Bensow R. Investigations on experimental and computational trim optimisation methods. Ocean Eng 2023;288:116098.
[155] Mahmoodi H, Ghamari I, Hajivand A, Mansoori M. A CFD investigation of the propulsion performance of a low-speed VLCC tanker at different initial trim angles. Ocean Eng 2023;275:114148.
[156] Fan X, Sun HS, Xiang XT. Investigation on trim optimization of a bulk carrier based on cfd method. Proc Int Offshore Polar Eng Conf 2020:3845–9.
[157] Shivachev E, Khorasanchi M, Day S, Turan O. Impact of trim on added resistance of KRISO container ship (KCS) in head waves: an experimental and numerical study. Ocean Eng 2020;211:107594.
[158] Li J, Duan W, Chen J, Ma S, Zhang Y. A study on dynamic trim optimization of VLCC oil tanker in wind and waves. Ocean Eng 2022;253:111270.
[159] Liu X, Wu J, Liao Y, Tu X. Column-based waterline detection for lightweight ship draft reading. IEEE Trans Instrum Meas 2022;71:1–10.
[160] Bisinotto GA, Sparano JV, Simos AN, Cozman FG, Ferreira MD, Tannuri EA. Sea state estimation based on the motion data of a moored FPSO using neural

networks: an evaluation with multiple draft conditions. Ocean Eng 2023;276:114235.
[161] De A, Choudhary A, Tiwari MK. Multiobjective approach for sustainable ship routing and scheduling with draft restrictions. IEEE Trans Eng Manag 2019;66(1):35–51.
[162] Liu S, Papanikolaou A. Approximation of the added resistance of ships with small draft or in ballast condition by empirical formula. Proc IME M J Eng Marit Environ 2019;233(1):27–40.
[163] Campbell R, Terziev M, Tezdogan T, Incecik A. Computational fluid dynamics predictions of draught and trim variations on ship resistance in confined waters. Ocean Eng 2022;126:103301.
[164] Bassam AM, Phillips AB, Turnock SR, Wilson PA. Artificial neural network based prediction of ship speed under operating conditions for operational optimization. Ocean Eng 2023;278:114613.
[165] Du Y, Meng Q, Wang S, Kuang H. Two-phase optimal solutions for ship speed and trim optimization over a voyage using voyage report data. Transp Res Part B Methodol 2019;112:88–114.
[166] Li M, Pan S, Cheng Y, Yuan Z, Tao L. Time-domain numerical simulation for multi-ships moving in waves with forward speed. Ocean Eng 2023;290:116325.
[167] Fan A, Yang J, Yang L, Liu W, Vladimir N. Joint optimisation for improving ship energy efficiency considering speed and trim control. Transport Res Transport Environ 2022;113:103527.
[168] Hao C, Lu Z, Feng Y, Bai H, Wen M, Wang T. Optimization of fuel/air mixing and combustion process in a heavy-duty diesel engine using fuel split device. Appl Therm Eng 2021;186:116458.
[169] Yousefi A, Guo H, Dev S, Liko B, Lafrance S. Effects of ammonia energy fraction and diesel injection timing on combustion and emissions of an ammonia/diesel dual-fuel engine. Fuel 2022;314:122723.
[170] Yan X, He Y, Fan A. Carbon footprint prediction considering the evolution of alternative fuels and cargo: a case study of Yangtze river ships. Renew Sustain Energy Rev 2023;173:113068.
[171] Xing H, Spence S, Chen H. A comprehensive review on countermeasures for CO_2 emissions from ships. Renew Sustain Energy Rev 2020;134:110222.
[172] Nadery A, Ghassemi H, Nowruzi H. Enhancement of the ship propeller hydrodynamic performance by different energy-saving devices mounted at the upstream zone. J Braz Soc Mech Sci Eng 2021;43:469.
[173] Gaggero S, Ferrando M. Wake instabilities of tip-loaded propellers: comparison between CLT and "new generation" CLT configurations. J Mar Sci Eng 2023;11(1):112.
[174] Li J, Zhao D, Wang C, Sun S, Ye L. Method for the calculation of the underwater effective wake field for propeller optimization. Water 2019;11(1):165.
[175] Windén B. Predicting the powering performance of different vessel types using an open-source CFD propulsion framework. SNAME Maritime Convention 2021. https://doi.org/10.5957/SMC-2021-132.
[176] Eom MJ, Jang YH, Paik KJ. A study on the propeller open water performance due to immersion depth and regular wave. Ocean Eng 2021;219:108265.
[177] Eom MJ, Paik KJ, Jang YH, Ha JY, Park DW. A method for predicting propeller performance considering ship motion in regular waves. Ocean Eng 2021;232:109135.
[178] Liu Z, Liu X, Wan D, Xu J. Ship resistance and wake optimization of jbc considering hull-propeller interaction based on cfd. Proceedings of the international offshore and polar engineering conference. 2021. p. 2549–56.
[179] Stan LC, Calimanescu I, Popa V. New innovative backflow marine propeller optimisation study by CFD. IOP Con Ser. Mater Sci Eng 2018;400:082018.
[180] Liu J, Zou Z, Guo H, Chen C. A study on the interaction among hull, engine and propeller during self-propulsion of a ship. Ocean Eng 2023;286:115702.
[181] Lovibond O, Elbarghthi AFA, Dvorak V, Wen C. Numerical analysis of propellers for electric boats using computational fluid dynamics modelling. Energy Convers Manag X 2023;17:100349.
[182] Su SC, Wang SS, Cao JB, Feng DS. Prediction of hydrodynamic characteristics of combined propellers based on CFD method. J Phys Conf 2019;1300:012016.
[183] Quist NA, Christensen RH, Mikkelsen H, Walther JH. Validation of a full scale CFD simulation of a self-propelled ship with measured hull roughness and effect of welding seams on hull resistance. Appl Ocean Res 2023;141:103746.
[184] Kim KW, Paik KJ, Lee JH, Song SS, Atlar M, Demirel YK. A study on the efficient numerical analysis for the prediction of full-scale propeller performance using CFD. Ocean Eng 2021;240:109931.
[185] Coraddu A, Dubbioso G, Mauro S, Viviani M. Analysis of twin screw ships' asymmetric propeller behaviour by means of free running model tests. Ocean Eng 2013;68:47–64.
[186] Franceschi A, Piaggio B, Villa D, Viviani M. Development and assessment of CFD methods to calculate propeller and hull impact on the rudder inflow for a twin-screw ship. Appl Ocean Res 2022;125:103227.
[187] Lu S, Cheng X, Liu J, Li S, Yasukawa H. Maneuvering modeling of a twin-propeller twin-rudder inland container vessel based on integrated CFD and empirical methods. Appl Ocean Res 2022;126:103261.
[188] Vimala V, Veeramanikandan K, Viswanathan N, Krishnan BP, Hari S, Madhavan S, Dhinesh S, Karthikeyan V. Analysis of marine propeller using computational fluid dynamics. Mater Today Proc 2022;68(6):2375–81.
[189] Acanfora M, Altosole M, Balsamo F, Micoli L, Campora U. Simulation modeling of a ship propulsion system in waves for control purposes. J Mar Sci Eng 2022;10(1):36.
[190] Guo H, Zou Z. CFD and system-based investigation on the turning maneuver of a twin-screw ship considering hull-engine-propeller interaction. Ocean Eng 2022;251:110893.
[191] Cui Y, Lam WH, Robinson D, Hamill G. Temporal and spatial scour caused by external and internal counter-rotating twin-propellers using Acoustic Doppler Velocimetry. Appl Ocean Res 2020;97:102093.
[192] Pereira FA, Capone A, Felice FD. Flow field and vortex interactions in the near wake of two counter-rotating propellers. Appl Ocean Res 2021;117:102918.
[193] Capone A, Felice FD, Pereira FA. On the flow field induced by two counter-rotating propellers at varying load conditions. Ocean Eng 2021;221:108322.
[194] Grassi D, Brizzolara S, Viviani M, Savio L, Caviglia S. Design and analysis of counter-rotating propellers-comparison of numerical and experimental results. Journal of Hydrodynamics, Ser. B 2010;22:570–6.
[195] Nowrouz MN, Mohammadi S, Zarezadeh M. Optimization of a marine contra-rotating propellers set. Ocean Eng 2018;167:397–404.
[196] Bai X, Ling H, Luo X, Li Y, Yang L, Kang J. Reliability and availability evaluation on hydraulic system of ship controllable pitch propeller based on evidence theory and dynamic Bayesian network. Ocean Eng 2023;276:114125.
[197] Guo B, Vartdal BJ, Øystein Åsheim A. CFD-based operation optimization of hybrid ships. Proceedings of the international conference on offshore mechanics and arctic engineering. OMAE; 2020.
[198] Zhu B, Xiao Q, Geng J. Research on a water-treading bionic propeller with variable pitch. Ocean Eng 2022;266:112989.
[199] Gypa L, Jansson M, Gustafsson R, Werner S, Bensow R. Controllable-pitch propeller design process for a wind-powered car-carrier optimising for total energy consumption. Ocean Eng 2023;269:113426.
[200] Ma X, Wang C. Comparative analysis of the efficiency of manual adjustable pitch propeller and fixed pitch propeller based on dual-scale coupling. J Phys Conf 2021;1865:032029.
[201] Yong Y, Motley MR, Barber R, et al. Adaptive composite marine propulsors and turbines: progress and challenges. Appl Mech Rev 2016;68(6):060803.
[202] An X, Wang P, Ye M, He R, Li C, Lessard L. Tip clearance influence on hydrodynamic performance and pressure fluctuation of a composite ducted propeller using a two-way FSI method. Ocean Eng 2023;282:114698.
[203] Joung TH, Sammut K, He F, Lee SK. Shape optimization of an autonomous underwater vehicle with a ducted propeller using computational fluid dynamics analysis. Int J Nav Archit Ocean Eng 2012;4(1):44–56.
[204] Li T, Sun S. Numerical study on the scale effects of two-dimensional cycloidal propellers. Ocean Eng 2023;279:114445.
[205] Cai BA, Tian BB, Qiu LY, Xu Q, Mao XF, He W, Chai W. Application of the body force method in the rim driven thruster. Int J Nav Archit Ocean Eng 2022;14:100476.
[206] Gaggero S. A study on the wake evolution of a set of RIM-driven thrusters. J Mar Sci Eng 2023;11(9):1659.
[207] Liu LL, Jiang W, Liu ZW, Yu RX, Bian T. Multi-objective optimization of shaftless rim-driven thruster based on ISIGHT. Journal of physics: conference series, the 5th international conference on mechanical, electric, and industrial engineering, vol. 2369; 2022, 012023.
[208] Yan X, Liang X, Ouyang W, Liu Z, Liu B, Lan J. A review of progress and applications of ship shaft-less rim-driven thrusters. Ocean Eng 2017;144:142–56.
[209] Jiang H, Ouyang W, Sheng C, Lan J, Bucknall B. Numerical investigation on hydrodynamic performance of a novel shaftless rim-driven counter-rotating thruster considering gap fluid. Appl Ocean Res 2022;118:102967.
[210] Zhai S, Jin SB, Chen JQ, Liu ZH, Song XL. CFD-based multi-objective optimization of the duct for a rim-driven thruster. Ocean Eng 2022;264:112467.
[211] Cai B, Xu Q, Tian B, Qiu L, Chai W, Qi J, He L. Improvement of the efficiency for rim-driven thrusters through acceleration of gap flow. Ocean Eng 2024;291:116480.
[212] Cao Q, Hong F, Tang D, Hu F, Lu L. Prediction of loading distribution and hydrodynamic measurements for propeller blades in a rim driven thruster. J Hydrodyn 2012;24(1):50–7.
[213] Yang Y. High-efficiency ship propeller based on electric direct drive without shaft rim. IOP Conf Ser Earth Environ Sci 2021;651:022068.
[214] Eslamdoost A. The hydrodynamics of waterjet/hull interaction. Chalmers university of technology; 2014.
[215] Jiao W, Wang Y, Cheng L, Zhao H. Experimental study on thrust pulsation characteristics of water jet propulsion pump units. Ocean Eng 2023;284:115079.
[216] Luo X, Li Q, Zhang Z, Zhang J. Research on the underwater noise radiation of high pressure water jet propulsion. Ocean Eng 2021;219:108438.
[217] Zhao D, Zhang Y, He Q, Sun C, Bi M. Hydrodynamic and flow field characteristics of water jet propulsion under mooring conditions. J Mar Sci Eng 2022;10(7):953.
[218] Eslamdoost A, Larsson L, Bensow R. Analysis of the thrust deduction in waterjet propulsion-The Froude number dependence. Ocean Eng 2018;152:100–12.
[219] Eslamdoost A, Larsson L, Bensow R. A pressure jump method for modeling waterjet/hull interaction. Ocean Eng 2014;88:120–30.
[220] Liu B, Xu X, Pan D, Wang X. Research on shipping energy-saving technology: hydrofoil amphibious vehicle driven by waterjet propulsion. J Clean Prod 2023;382:135257.
[221] Shirazi AT, Nazari MR, Manshadi MD. Numerical and experimental investigation of the fluid flow on a full-scale pump jet thruster. Ocean Eng 2019;182:527–39.
[222] Lee JH, Lee JH, Kim MM, Oh D, Paik KJ. A numerical study on the optimization of the slit shape of a jet injection propeller. Int J Nav Archit Ocean Eng 2024;16:100578.
[223] Wang Z, Min S, Peng F, Shen X. Comparison of self-propulsion performance between vessels with single-screw propulsion and hybrid contra-rotating podded propulsion. Ocean Eng 2021;232:109095.
[224] Pruszko H, Reichel M, Necel M, Brüns S. Towards a reliable method for extrapolation of propulsion performance for vessels with twin-crp-pod system. Ocean Eng 2024;310(2):118758.

[225] Wang Z, Xiong Y, Wang R, Zhong C. Numerical investigation of the scale effect of hydrodynamic performance of the hybrid CRP pod propulsion system. Appl Ocean Res 2016;54:26–38.
[226] Kim S, Kinnas SA. Numerical prediction of underwater noise on a flat hull induced by twin or podded propeller systems. J Sound Vib 2022;539:117256.
[227] Zhang YX, Cheng XK, Feng L. Numerical investigation of the unsteady flow of a hybrid CRP pod propulsion system at behind-hull condition. Int J Nav Archit Ocean Eng 2020;12:918–27.
[228] Wu R, Zhang C, Chen X, Wang Z, Yan X. Influence analysis of podded propulsors installation position based on self propulsion simulation. 2023 7th international conference on transportation information and safety (ICTIS). 2023. p. 1301–7. Xi'an, China.
[229] Park HG, Choi JK, Kim HT. An estimation method of full scale performance for pulling type podded propellers. Int J Nav Archit Ocean Eng 2014;6(4):965–80.
[230] Choi JK, Park HG, Kim HT. A numerical study of scale effects on performance of a tractor type podded propeller. Int J Nav Archit Ocean Eng 2014;6(2):380–91.
[231] Zhou X, Li T, Wang N, Wang X, Chen R, Li S. Pilot diesel-ignited ammonia dual fuel low-speed marine engines: a comparative analysis of ammonia premixed and high-pressure spray combustion modes with CFD simulation. Renew Sustain Energy Rev 2023;173:113108.
[232] Xu L, Xu S, Bai X, Repo JA, Hautala S, Hyvönen J. Performance and emission characteristics of an ammonia/diesel dual-fuel marine engine. Renew Sustain Energy Rev 2023;185:113631.
[233] Hussain M, Abdel-Nasser Y, Banawan A, Ahmed YM. FSI-based structural optimization of thin bladed composite propellers. Alex Eng J 2020;59(5):3755–66.
[234] Xu LJ, Hu H, Ji J, Cai JY, Dai LY. Hybrid energy saving performance of translucent CdTe photovoltaic window on small ship under sailing condition. Energy 2024;295:131070.
[235] Hino T, Hirata N, Ohashi K, Toda Y, Zhu T, Makino K, Takai M, Nishigaki M, Kimura K, Anda M, Shingo S. Hull form design and flow measurements of a bulk carrier with an energy-saving device for CFD validations. Prads 2016 - proceedings of the 13th international symposium on PRActical design of ships and other floating structures. 2016.
[236] Windén B. An open-source framework for ship performance CFD. SNAME 26th Offshore Symposium, TOS 2021. https://doi.org/10.5957/tos-2021-22.
[237] Díaz Ojeda HR, Oyuela S, Sosa R, Otero AD, Pérez Arribas F. Fishing vessel bulbous bow hydrodynamics—a numerical reverse design approach. J Mar Sci Eng 2024;12(3):436.
[238] Stark C, Xu YX, Zhang M, Yuan ZM, Tao LB, Shi WC. Study on applicability of energy-saving devices to hydrogen fuel cell-powered ships. J Mar Sci Eng 2022;10(3):388.
[239] Dong XL. Ship energy saving and emission reduction. E3S Web of Conferences 2020;165:01017.
[240] Shen H, Obwogi E, Su Y. Scale effects for rudder bulb and rudder thrust fin on propulsive efficiency based on computational fluid dynamics. Ocean Eng 2016;117:199–209.
[241] Wu PC, Chang CW, Huang YC. Design of energy-saving duct for JBC to reduce ship resistance by CFD method. Energies 2022;15(17):6484.
[242] Bakica A, Vladimir N, Jasak H, Kim ES. Numerical simulations of hydrodynamic loads and structural responses of a Pre-Swirl Stator. Int J Nav Archit Ocean Eng 2021;13:804–16.
[243] Obwogi EO, Shen H, Su Y. The design and energy saving effect prediction of rudder-bulb-fin device based on CFD and model test. Appl Ocean Res 2021;114:102814.
[244] Zhu J, Li C. A probabilistic multi-objective design method of sail-photovoltaic-hybrid power system for an unmanned ocean surveillance trimaran. Appl Energy 2023;350:121604.
[245] He J, Hu Y, Tang J, Xue S. Research on sail aerodynamics performance and sail-assisted ship stability. J Wind Eng Ind Aerod 2015;146:81–9.
[246] Munro MZ, Mizzi K, Gutteridge K, Atlar M, Sasaki N. The full-scale performance prediction of a general cargo SHIP with a retrofitted gate rudder system using CFD procedures. IET Conference Proceedings 2023;7:315–22.
[247] Tacar Z, Sasaki N, Atlar M, Korkut E. An investigation into effects of Gate Rudder® system on ship performance as a novel energy-saving and manoeuvring device. Ocean Eng 2020;218:108250.
[248] Gopinath S, Vijayakumar R. Computational analysis of the effect of hull vane on hydrodynamic performance of a medium-speed vessel. J Mar Sci Appl 2023;22:762–74.
[249] Soma GC, Vijayakumar R. Numerical investigation on the effect of hull vane for a high-speed displacement vessel. Ships Offshore Struct 2023:1–14.
[250] Nguyen VT, Chandar D. Ship specific optimization of pre-duct energy saving devices using reduced order models. Appl Ocean Res 2023;131:103449.
[251] Zalachoris A, Themelis N. Assessment of the efficiency of a Mewis duct through performance analysis of operational data. J Mar Sci Appl 2022;21:144–58.
[252] Ma Y, Bi H, Hu M, Zheng Y, Gan L. Hard sail optimization and energy efficiency enhancement for sail-assisted vessel. Ocean Eng 2019;173:687–99.
[253] Cairns J, Vezza M, Green R, MacVicar D. Numerical optimisation of a ship wind-assisted propulsion system using blowing and suction over a range of wind conditions. Ocean Eng 2021;240:109903.
[254] Bordogna G, Muggiasca S, Giappino S, Belloli M, Keuning JA, Huijsmans RHM. The effects of the aerodynamic interaction on the performance of two Flettner rotors. J Wind Eng Ind Aerod 2020;196:104024.
[255] Copuroglu HS, Pesman E. Analysis of Flettner Rotor ships in beam waves. Ocean Eng 2018;150:352–62.
[256] Gürkan AY, Ünal UO, Aktaş B, Atlar M. An investigation into the gate rudder system design for propulsive performance using design of experiment method. Ship Technol Res 2023;71(2):199–212.
[257] Makram TE, Panagiotou P, Mattheou D. Wingsail layout design and shape optimization using a CFD-aided Taguchi approach: the Aegean Marathon case study. Ocean Eng 2023;276:114055.
[258] Bigi N, Roncin K, Leroux JB, Parlier Y. Ship towed by kite: investigation of the dynamic coupling. J Mar Sci Eng 2020;8(7):486.
[259] Deng Y, Zhang X, Zhang G, Huang C. Parallel guidance and event-triggered robust fuzzy control for path following of autonomous wing-sailed catamaran. Ocean Eng 2019;190:106442.
[260] Ebrahimi M, Jahangirian A. Aerodynamic optimization of airfoils using adaptive parameterization and genetic algorithm. J Optim Theor Appl 2014;162:257–71.
[261] Wang K, Xue Y, Xu H, Huang L, Ma R, Zhang P, Jiang X, Yuan Y, Negenborn RR, Sun P. Joint energy consumption optimization method for wing-diesel engine-powered hybrid ships towards a more energy-efficient shipping. Energy 2022;245:123155.
[262] Kiryanto, Budiarto U, Firdhaus A. Analysis of the effect of hull vane on ship resistance using CFD methods. IOP Conf Ser Earth Environ Sci 2021;649:012051.
[263] Soma GC, Vijayakumar R. Hydrodynamic performance of high-speed displacement vessel with hull vane. Ocean Eng 2023;285(1):115362.
[264] Atlar M, Aktas B, Gurkan A, Sasaki N, Sun X, Korkut E, Felli M. The GATERS Project – an innovative way of retrofitting ships for greener and safer operations. Transport Res Procedia 2023;72:1958–65.
[265] Munazid A, Munazid A, Ariana IM, Ikap Utamaet. Numerical study of pre-duct on traditional fishing vessels as energy saving device (ESD). IOP Conf Ser Earth Environ Sci 2020;557:012049.
[266] Trimulyono A, Mulyatno IP, Rachmat AF. The effect of Mewis duct energy saving device to propeller performance. IOP Conf Ser Earth Environ Sci 2022;1908:012076.
[267] Shen HL, Obwogi EO, Su YM. The design and energy saving effect of partial duct and unconventional pre-swirl fin and their combination system. Ocean Eng 2023;287(1):115857.
[268] Tan Q, Ding Y, Liu J, Sui C. CFD-based hull-engine-propeller matching study after retrofitting propeller boss-cap-fin (PBCF). Ocean Eng 2023;285(1):115364.
[269] Zhang R, Huang L, Peng G, Ma R, Wang K, Tian F, Song Q. A novel method of desynchronized operation of sails for ship wind-assisted propulsion system. Ocean Eng 2023;288:115964.
[270] Hasselaar TWF, Xing-Kaeding Y. Evaluation of an energy saving device via validation speed/power trials and full scale CFD investigation. Int Shipbuild Prog 2017;63(3–4):169–95.
[271] Zeng Q, Zhang X, Cai W, Zhou Y. Wake distortion analysis of a Dynarig and its application in a sail array design. Ocean Eng 2023;278:114341.
[272] Mandruand A, Pacuraru F. The effect of appendages on ship resistance. Conference Series: Mater Sci Eng 2021;1182:012041.
[273] Liu Z, Liu W, Chen Q, Luo F, Zhai S. Resistance reduction technology research of high speed ships based on a new type of bow appendage. Ocean Eng 2020;206:107246.
[274] Yin C, Rosenvinge CK, Sandland MP, Ehlers A, Shin KW. Improve ship propeller efficiency via optimum design of propeller boss cap fins. Energies 2023;16:1247.
[275] Li D, Zhang Y, Dai J, Li G. Aerodynamic performance of a new double-flap wing sail. Pol Marit Res 2019;26(4):61–8.
[276] Sang Y, Ding Y, Xu J, Sui C. Ship voyage optimization based on fuel consumption under various operational conditions. Fuel 2023;352:129086.
[277] Vigna V, Figari M. Wind-assisted ship propulsion: matching flettner rotors with diesel engines and controllable pitch propellers. J Mar Sci Eng 2023;11:1072.
[278] Le TK, He NV, Hien NV, Bui NT. Effects of a bulbous bow shape on added resistance acting on the hull of a ship in regular head wave. J Mar Sci Eng 2021;9(6):559.
[279] Zhang X, Wang J, Wan D. Numerical techniques for coupling hydrodynamic problems in ship and ocean engineering. J Hydrodyn 2020;32:212–33.
[280] Jiang J, Cai H, Ma C, Qian Z, Chen K, Wu P. A ship propeller design methodology of multi-objective optimization considering fluid–structure interaction. Engineering Applications of Computational Fluid Mechanics 2018;12(1):28–40.
[281] Guan G, Zhang X, Wang P, Yang Q. Multi-objective optimization design method of marine propeller based on fluid-structure interaction. Ocean Eng 2022;252:111222.
[282] Huang Q, Yan X, Zhang C. Numerical calculation and experimental research on the ship dynamics of the fluid-structure interaction. Adv Mech Eng 2018;10(7).
[283] Tu H, Chen H. Numerical investigation of interaction between propulsion system behaviour and manoeuvrability of a large containership. Proc IME M J Eng Marit Environ 2022;236(2):355–75.
[284] Huang Z, Chen Z, Zhang Y, Xiong Y, Duan K. The scale effect study on the transient fluid–structure coupling performance of composite propellers. J Mar Sci Eng 2022;10(11):1725.
[285] Zhang X, Hong Y, Yang F, Jiao W, Wang R. Effect of rudder on propulsion performance and structural deformation of composite propellers. Ocean Eng 2019;182:318–28.
[286] Zhang LF, Hou LF, Tao YH. Analyzing the performance of hull and water jet propeller under mooring conditions: effects of shallow water depths and varying inflow speeds. Eng Comput 2024;41(3):755–92.
[287] Badoe C, Phillips A, Turnock S. Influence of drift angle on the computation of hull–propeller–rudder interaction. Ocean Eng 2015;103:64–77.
[288] Taskar B, Yum KK, Steen S, Pedersen E. The effect of waves on engine-propeller dynamics and propulsion performance of ships. Ocean Eng 2016;122:262–77.

[289] Liu J, Guo H, Zou Z, Liu Y. A study on interaction among hull, diesel engine and propellers of a twin-screw ship during turning circle maneuver. Appl Ocean Res 2023;185:103517.

[290] Liu J, Gu Z, Liu S. Research on MDO of ship propulsion shafting dynamics considering the coupling effect of a propeller-shafting-hull system. Pol Marit Res 2023;30(1):86–97.

[291] Song K, Guo C, Wang C, Sun C, Li P, Wang W. Numerical analysis of the effects of stern flaps on ship resistance and propulsion performance. Ocean Eng 2019;193:106621.

[292] Garnier P, Viquerat J, Rabault J, Larcher A, Kuhnle A, Hachem E. A review on deep reinforcement learning for fluid mechanics. Computers & Fluids 2021;225:104973.

[293] Liu L, Yang Y, Peng T. Machine learning prediction of 6-DOF motions of KVLCC2 ship based on RC model. J Ocean Eng Sci 2022. https://doi.org/10.1016/j.joes.2022.08.004.

[294] Lu Y, Yuan JP, Si QR, Ji PF, Tian D, Liu JF. Study on the optimal design of a shark-like shape AUV based on the CFD method. J Mar Sci Eng 2023;11(10):1869.

[295] Yang Y, Zhang Z, Zhao J, Zhang B, Zhang L, Hu Q, Sun J. Research on ship resistance prediction using machine learning with different samples. J Mar Sci Eng 2024;12:556.

[296] Fahrnholz SF, Caprace JD. A machine learning approach to improve sailboat resistance prediction. Ocean Eng 2022;257:111642.

[297] Bagazinski NJ, Ahmed F. Ship-D: ship hull dataset for design optimization using machine learning. Proceedings of the ASME 2023 international design engineering technical conferences and computers and information in engineering conference, 49th design automation conference (DAC), vol. 3; 2023 (A).

[298] Zhang SL, Tezdogan T, Zhang BJ, Lin L. Research on the hull form optimization using the surrogate models. Engineering Applications of Computational Fluid Mechanics 2021;15(1):747–61.

[299] Ao Y, Duan HL, Li SF. An integrated-hull design assisted by artificial intelligence-aided design method. Comput Struct 2024;297:107320.

[300] Li L, Chen YH, Qiang YM, Zhou B, Chen WZ. Construction and application of numerical diagram for high-skew propeller based on machine learning. Ocean Eng 2023;278:114480.

[301] Zarezadeh M, Nouri NM, Madoliat R, Alagheband SHS. Optimal section design of surface piercing propeller using artificial neural networks algorithm. Ships Offshore Struct 2024:1–11.

[302] Lee MK, Lee I. Optimal design of flow control fins for a small container ship based on machine learning. J Mar Sci Eng 2023;11(6):1149.

[303] Vasilev M, Kalajdžić M, Ivković I. CFD-powered ship trim optimization: integrating ANN for user-friendly software tool development. J Mar Sci Eng 2024;12(8):1265.

[304] Liu M, Song Y, Yang S, Yang M. Rapid data-driven individualised shape design of AUVs based on CFD and machine learning. Ships Offshore Struct 2024:1–15.

[305] Usman A, Rafiq M, Saeed M, et al. Machine learning computational fluid dynamics. 2021 Swedish artificial intelligence society workshop (SAIS). IEEE; 2021. p. 1–4.

[306] La Ferlita A, Qi Y, Di Nardo E, El Moctar O, Schellin TE, Ciaramella A. A framework of a data-driven model for ship performance. Ocean Eng 2024;309(1):118486.

[307] Choi YS, Hong SY, Song JH. Composite propeller design optimization for cavitation minimization using deep learning-based objective parameter prediction model. Ocean Eng 2023;287(1):115760.