Research Status and Analysis of the Ship Hydrodynamic Energy-saving Devices

*LuRong Xu, Dongya He, Decheng Wan**

State Key Laboratory of Ocean Engineering, School of Naval Architecture, Ocean and Civil Engineering, Shanghai Jiao Tong University, Collaborative Innovation Center for Advanced Ship and Deep-Sea Exploration, Shanghai, China

*Corresponding author

Abstract: In the nowadays shipping market, the energy-saving method of the ship has become a hot subject. As one of the effective ways, hydrodynamic energy-saving devices(ESD) has the advantages of low cost, satisfactory energy-saving effect and the convenient transformation on the old ships. Traditionally, the method of model test is applied in the study of the ESDs, but is has the disadvantages of high cost and long research period. Along with the development of the computer technology, computational fluid dynamics(CFD) is used more and more frequently in the study of ESDs.

This paper has summarized the research status of the hydrodynamic ESDs using CFD in the recent years. The ESDs were divided into four kinds according to their working principle: ESDs based on inflow optimization technology of propeller, ESDs based on energy recovery technology of propeller, ESDs based on the reduction or separation of the vortex induced by the ship and new propellers. For each kind of ESD, the paper has done some introduction as well as its working principle and then, the study achievements of ESDs with CFD method were summarized. At last, the development of the CFD study on the ESDs were prospected.

Introduction

With the problems of the energy shortage, environmental pollution, greenhouse effect in the recent years, as a new wind, design concept for Green Ship gradually attracted the attention of the relevant personnel in the industry. For a long time, the shipping market has been an industry of energy consumption, and a cause of series of environmental problems such as greenhouse effect. In early 2011, the 62nd meeting of the MEPC, the Annex VI amendment of MARPOL 73 /78 Convention was adopted formally which determined the EEDI and SEEMP ship energy efficiency standards. These two criteria shall be effective on January 1, 2013 and enforced in 2015[1]. New design ships with exceeded EEDI index will not be audited by the classification society or IMO. The operational ship with EEDI exceeded index may be forced to scrap or not enter the international shipping market. Therefore, energy saving and emission reduction becomes an important problem that needs to be paid more attention to. The ship hydrodynamic energy saving device has a significant effect, and has important influence on the study of the energy saving technology of the ship.

According to the different working principles, the hydrodynamic ESD can be divided into four categories: the first kind of ESDs is based on inflow optimization technology of propeller which can improve the inflow of the propeller, optimize the wake in the stern. This kind of ESDs mainly includes the wake equalizing duct, the pre-swirl stator and the energy-saving shaft bracket. The second kind of ESDs is based on energy recovery technology of propeller which can recovery the rotation energy of the wake of the propeller. This kind of energy saving device mainly includes the contra-rotating propeller, the rudder ball, the rudder attached thrust fin, the free rotating impeller wheel and the reaction rudder. The third kind of ESDs is based on the reduction or separation of the vortex induced by the ship[2]. The representative device is mainly Propeller Boss Cap Fins (PBCF) . The fourth kind of ESDs is new propeller which mainly include CRPs (Contra-Rotating Propellers), sharp fin propeller and bionic thruster.

Traditionally, model tests or theoretical research is the main method in the study of the ESDs. In the recent years, CFD (Computational Fluid Dynamics) is developed quickly. As a new means in the fluid mechanics, CFD is a satisfactory in that it can greatly short the design time and save the cost. Compared to the model test, CFD is almost never limited by the size or velocity of the ships and has less limitation and more application range than theoretical research. What is more, computer simulation will not affect the fluid of the flow which may make the phenomenon of the flow become subtle.[4] Many universities and institutes has done continuous research on the ESDs. For example, CMHL team led by Professor Decheng Wan in shanghai Jiao Tong University has developed a CFD solver naoe-FOAM-SJTU based on the open software OpenFOAM. Since this solver used the method overset grid, it can deal with the complex flow problems of ship-propeller-rudder multilevel and multi degree of freedom well. Also, the resistance performance and maneuver performance of the ships can also be solved with this solver[3]. Now, naoe-FOAM-SJTU solver has been used in the study of the influence of ESDs on the propulsion performance of propellers.

1. Energy-saving devices based on inflow optimization technology of propeller

1.1 Wake equalizing duct

Wake equalizing duct (WED) is firstly applied in the bulk carrier by German hydrodynamics professor Schneekluth in 1980s.[5] This device consist of two semi-rings is installed in the above the shaft of the propeller in the stern of ship. It is called as “fore duct” or “compensating duct”. The function it acts is as the followings.

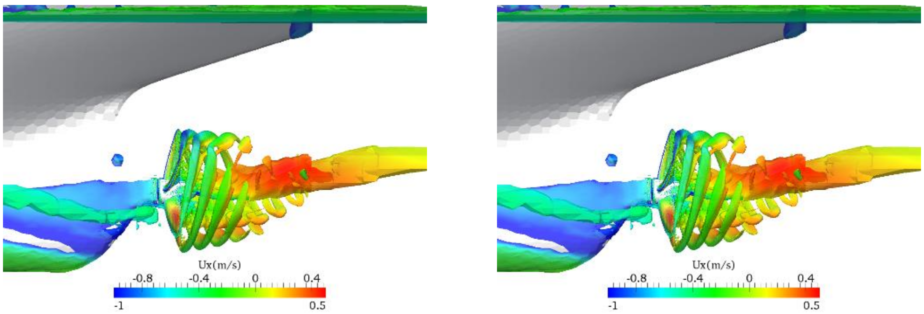
(1) It can improve the inflow velocity above the propeller, make the inflow on the face of the propellers uniform and improve the propulsion effencient.

(2) It can reduce the flow separation in the stern and the resistance when the ship is sailing.

(3) It can provide extra thrust for the ship if the duct is installed in the appropriate positon and angle.

Fig. 1 Wake equalizing duct

Fahri Celik[6] used commercial software FLUENT to do the numerical simulation of the duct on a carrier. The calculation used RANS to obtain the detailed wake field. The paper focused on the analysis of the wake and the propulsion performance of the propeller while the carrier was at the speed of 10~16 knots. The results of the calculation showed that the WED could effectively improve the propulsion and reduce the viscous resistance if a ship had a large block coefficient. Also, it indicated that the installation of the WED had a great influence on the energy-saving effect. Heinke[7] used commercial software CFX to study the scale effect of the WED installed on the typical container ship. The tetrahedral mesh and hexahedral mesh were used in the simulation and the resistance of the hull, propulsion performance and the flow field were calculated in both the real ship and the ship model with or without the WED. Chonghong Yin[8] of CMHL conducted the numerical simulation on the JBC with the WED using naoe-FOAM-SJTU solver which had the module of the overset grid based on the RANS equation. This solver calculated the thrust and the torque of the propeller in each advanced coefficient in one time using Single-Run method. The results showed that the WED can improve the propulsion performance of the JBC, and the energy-saving effect was about 6.7%. fig. 3 is the wake vortex.



(a) JBC without the WED (b) JBC with the WED

Fig. 2 the wake vortex of the self-propulsion ship[8]

Martinas[9] done the parameter optimization for a certain ship with the WED. This paper used CFD method to study the effect of the installation angle, the inlet and outlet radius, the length between the duct and the propeller on the energy-saving effect. Because of limit of the computational ability, only the stern of the model was considered while the free surface was not in the paper. Through large amount of the numerical calculation, the design was obtained. Shaofeng Huang[10] conducted the study on the scale effect and the energy-saving principle of the duct installed in the bulk carrier. This paper used the method that solved 3-dimensional incompressible RANS equation and ignored the influence of the free surface. The paper drew two conclusions at last: the WED could reduce the loss of the rotation energy in the stern and produce extra thrust; along with the increase of the Reynold number, the extra thrust became bigger and the loss of the energy decreased. Therefore, in the real ship, the WED could obtain better energy-saving effect. Tao Sun[11] of CMHL studied the effect of the inlet and outlet diameter and the length of the duct on the energy-saving effect using naoe-FOAM-SJTU solver. The calculation used k-ωSST turbulence model and overset grid. Comparing the resistance of the JBC with and without the WED, the paper showed that to make the WED had the ability of rectification and accelerating the inflow, the outlet diameter should smaller than the inlet diameter. If the outlet diameter was too small, the energy-saving effect of the WED would reduce a lot. While the proportion of the inlet and out let diameter remained unchanged, the energy-saving effect of the WED would decrease if the inlet and outlet diameter changed. Hailong Shen[12] prospected a method to forecast the energy-saving effect of the WED.

This method was based on the surface element method and RANS equation. The paper prospected the wake field of a ship with the WED at the speed of 15 knots. The result showed that this method could be applied in the forecast of the performance of the WED and the other ESDs.

1.2 Pre-swirl stator

Pre-swirl stator is a kind of energy-saving device which is installed before the propeller in the stern which is consist of several stators. It can make the inflow pre-rotate and recycle the rotation energy loss of the propeller to achieve the aim of reducing the energy consumption.

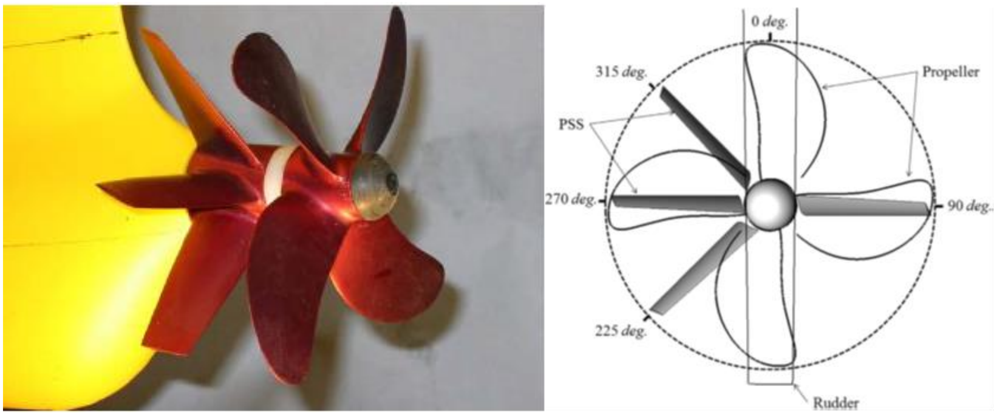


Fig 3 Pre-swirl stator

Sunho Park[13] used commercial software STAR-CCM+ to study the energy-saving effect of the pre-swirl stator (PSS) installed on the KVLCC2. In the calculation, the influence of the free surface was ignored. The author put forward a new method to obtain the reliable propulsion performance forecast from the ship models. Specifically, the author used the wake friction obtained from solving RANS equation subtracted that from the Euler equation. The result of the subtraction was seen as the viscous wake friction and can be extended to the real ship. Comparing the extended result and the direct result from the full scale calculation, this paper proved that this method could extend the energy-saving effect obtained from the ship model to the real ship. Naijun Lin[14] et al used the CFD method to analyze the influence of the designed circulation of the PSS installed on a 7 tones bulk carrier on the energy-saving effect. The weighting factor of the circulation was introduced in the calculation with lifting line theory used. The calculation result from the CFD was compared to that from the tank test and the conclusion that the PSS could have about 4.54% energy-saving effect was drawn. Dang[15] used a piece of code named ReFRESCO to study the detailed flow field of the propulsion system with the PSS using PIV and CFD two methods. It drew the conclusion that the CFD method had the ability to catch the detail and possible separation of the flow. The calculation result from the CFD method was the same as that from the measurement of the PIV. It indicated the reliable of the CFD method. Keunjae Kim[16] used the CFD commercial software SHIPFLOW to study the design and optimization for the PSS in the early design stage of a Aframax tanker. Yong-Jin Shin[17] used the CFD commercial software STAR CCM+ to analyze the resistance of a 3600 TEU container ship with the asymmetric PSS. In the numerical simulation, the incompressible RANS equation and k-ε turbulence were used. The calculation result showed that while at the design speed (24 knots), the propulsion efficient of the ship with the PSS would higher than the ship without the PSS about 6.65%. The CFD result only had a fine difference with the test result which indicated the reliable of the CFD. Yan Xing Kaeding[18] used the program FreSCo+ based on the RANS equation and the program QCM based on the calculation of the propeller to study the complex interaction among hull, ESD and propeller. It drew the conclusion that the energy-saving effect calculated from the CFD was 5.3% while the measured result from the sea trial was about 6.8%.

1.3 Energy-saving shaft strut

Energy-saving shaft strut was formed by two support arms. The steady blade above the propeller could make the flow have a circulation which was opposite with the rotation of the propeller before the flow entered into the propeller disk. This could counteract the circular induced velocity which could improve the propulsion performance.

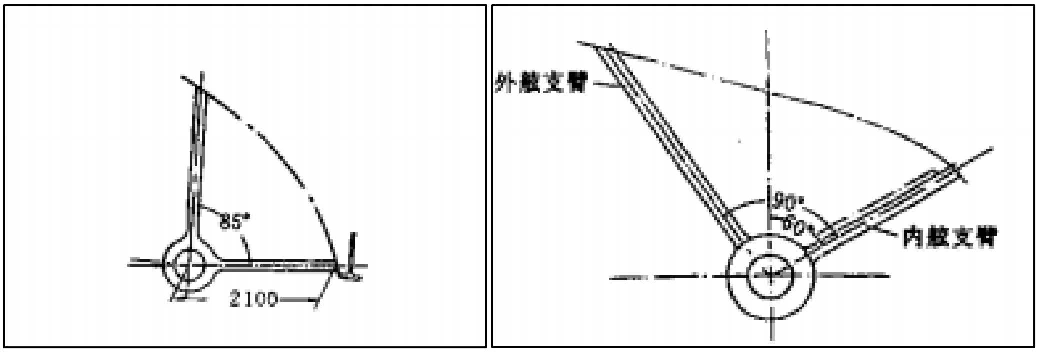


Fig. 4 Energy-saving shaft strut

Zhanzhi Wang[19] conducted the numerical simulation on the 3-diamention viscos flow field of the ship with appendage using RANS method. This paper focused on the section form of the shaft strut and on the installation angel and studied what the two factors had on the ship viscos resistance and wake field. It found that it was practical to determine the installation angle according to the direction of the velocity vector of the field before the strut. Keqiang Chen done the design on the strut of a twin-screw ship using the CFD method based on the potential flow theory and the energy-saving effect of that was calculated.

2. Energy-saving devices based on energy recovery technology of propeller

2.1 Rudder ball

Rudder ball is a kind of ESDs which is installed on the rudder blade at the center position of the propeller shaft. Its working principle is shown as the followings.

(1) The rudder ball can effectively cut down the space of low pressure at the axis after the propeller.

(2) The rudder ball can strengthen the rectification of the rudder and reduce the circumferential velocity of propeller.

(3) The rudder ball can make the wake field which is at the propeller disk more uniform and is beneficial to the cavitation performance and exciting performance of the propeller.

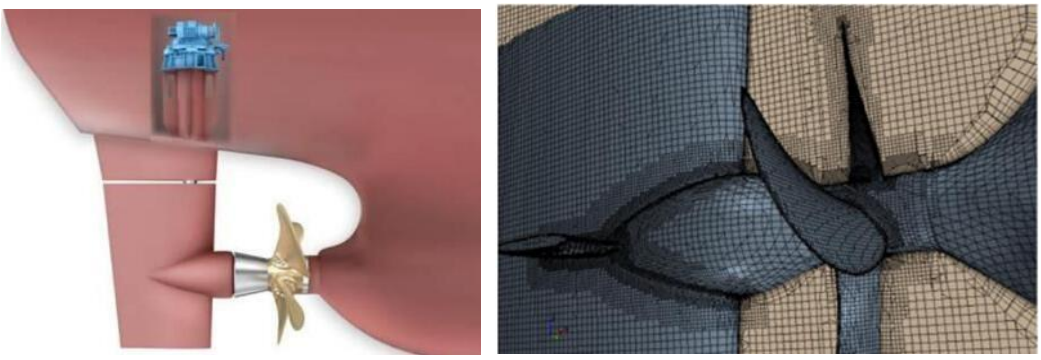


Fig. 5 the rudder ball and the grid

Guangzhou Yan[20] used the CFD commercial software to analyze the efficient of the rudder ball and the fin stabilizer under open water condition. The target ship was a container ship with rudder ball or the fin stabilizer. The calculation result showed that the combination using of the rudder ball and the fin stabilizer could bring about better energy-saving effect. While the fin stability had an obvious scale effect, the rudder ball had a very weak scale effect. Hailong Shen[21] studied a new kind rudder ball whose diameter was added to 0.3 time of the propeller diameter. The rudder ball could obtain the best energy-saving effect if it was installed at the axis of the propeller. Through large amount of the numerical simulation, this paper found that the rudder ball could have about 2.03% of efficient. Guibiao Wang[22] designed a new kind rudder ball by using the MRF method provided by the CFD commercial software FLUENT and the RNG k-ε turbulence model. The hydrodynamic performance of the propeller-rudder-rudder-ball system was analyzed. The research target was a 42 meters long trawler with duct rudder. It drew the conclusion that because of the exist of the duct, the energy-saving effect of the rudder ball would increase first, then decrease, then increase again and then decrease along with the increase of the advance coefficients. The curve of the energy-saving effect would show two peaks and one trough. Xuejun Li[23] et al, used the commercial CFD software FLUENT to do the numerical simulation on the AU5_593 propeller. This paper used the large eddy simulation and 3-dimension incompressible RANS equation. The interaction between the rudder and the propeller as well as the influence of the size of the rudder ball on the propeller efficient. It drew that the ratio between the bulk ball and the propeller existed a best number. While the ratio was around the best number, the bulk ball can bring its superiority into full play. Leiqiang Chen[24] used the CFD commercial software FLUENT to compare the ship model with or without the bulk ball. The research target was an oil tanker. It drew the conclusion that for each diameter of the propeller, there remained a best diameter for the bulk ball where the bulk ball could show the best energy-saving efficient. In addition, while the gap between the propeller and the bulk ball was determined, the longer the bulk ball was, the more thrust it would provide.

2.2 Thrust fin

Japanese corporation IHI designed a new ship ESD——thrust fin which was installed on the two sides of the rudder and has an opposite installation angle to the direction of the inflow. It could it turned the rotation energy of flow after the propeller into the thrust. Its working principle was that when the propeller rotated, the thrust fin had an excursion, the thrust would be produced for the interaction between the fin blade and the wake flow. The followings are two factors that can affect the energy-saving effect of the thrust fin.[25]

Fig. 6 Thrust fin

(1) The installation angle of the thrust fin. According to different ships, there existed a best installation angle. Generally speaking, this angle was about 5°.

(2) The length of the thrust fin. While the wingspan of the thrust fin was among the wake flow field, the thrust was produced for the interaction between the flow and the fin blade; while the wingspan was beyond the range of the field, the thrust would decrease along with the decrease of the propulsion efficient.

Wenquan Wang[26] conducted the CFD simulation of self-propulsion with rolling and pitching for a 75000 tones ship model. The commercial software STAR CCM+ was used to calculate the hydrodynamic numerical result of the ship-propeller-rudder system with or without thrust fin. It drew the conclusion that the thrust fin was beneficial to the increase of the propeller efficient in the hydrostatic test and the motion test. Haizhou Hu[27] et al, used the CFD method to calculate the energy-saving efficient the thrust fin brought about on a bulk carrier. In the calculation, the VOF model was used to simulate the gas-liquid two phase flow and the turbulence model was k-ε model. The rotation of the propeller was achieved by the MRF model. It drew the conclusion that the thrust fin could absorb the wake energy, increase the thrust of the propeller, reduce the resistance of the hull and bring about 2.3%energy-saving efficient. Wentin Hu[28] used the CFD commercial software FLUENT to forecast the hydrodynamic performance of a propeller with the rudder ball and the thrust fin. The RNG k-ε was used to simulate the real flow field. Hailong Sheng[29] used the CFD commercial software STAR CCM+ to study the energy-saving effect of the thrust fin installed on a 35000 tones bulk carrier. The calculation used RANS method and the VOF model to simulate the gas-liquid two phase flow. K-ε turbulence model was also used. The paper studied the scale effect of the thrust fin. Sing-Kwan Lee et al[30], used the CFD method to study performance of a container ship with thrust fin while its bow or stern was crashed.

2.3 Grim vane wheel

Grim vane wheel can turn the rotation energy of the water after the propeller into the thrust that is useful for the ship moving. The design of it originated in 1960s. The grim cane wheel can be installed during the building of the new ship or it can also be installed on the old ship conveniently[31].

Its working principle was as the followings.

(1) The wake flow of the propeller can drive the blades of turbine shape rotate so that the blades of the wing turn the rotation energy into the additional thrust.

(2) The grim vane wheel can reduce pulsating pressure on the hull about 40%~50% so that the shaking of the hull is weaken.

Fig. 7 Grim vane wheel

Lixun Hou[32] et al, used propeller vortex theory to conduct the design of the grim vane wheel. The surface element method was used to calculate shrinking percentage of the wake flow of the propeller so that the turbine part of the wheel can be determined. Also, the hydrodynamic performance of the propeller and the grim vane wheel were also calculated. It drew the conclusion that the bigger the advanced coefficient was and the small the area-ratio was, the energy-saving of the wheel was.

3. Energy-saving devices based on the reduction or separation of the vortex induced by the ship

3.1 Propeller Boss Cap Fins

Propeller Boss Cap Fins (PBCF) is a kind of ESDs which is installed after the propeller and rotate along with the propeller. Since the invention of the PBCF, it is used wide for its good performance.[33]-[36]

PBCF can recycle the energy of the rotational wake flow of the propeller root. Its energy-saving principle can be summarized as the followings.

(1) The blades of the PBCF has the ability of rectification which can eliminate the propeller vortex so that the pressure of the boss and the cap can be recovered and the thrust of the propeller is increased.

(2) The blade of the PBCF can provide the opposite torque so that reduce the whole torque of the propeller.

Fig. 8 Propeller Boss Cap Fins

Kurt et al[37], used the CFD commercial software STAR-CCM+ to do the design and the optimization of the PBCF. This paper studied a series of parameters which had effect on the energy-saving of the PBCF. The general design stage of the design and the optimization were summarized. From the calculation, it drew that the PBCF had about 1.3% energy-saving efficient and increased the energy using ratio. Xiaoyong Shi[38] used commercial software FLUNENT to study the performance of the PBCF installed on a82000DWT bulk carrier. This paper focused on the influence of the installation angle on the its energy-saving effect. The calculation result showed that the PBCF had an appropriate installation angle among which the PBCF can have about 1.75% energy-saving effect. Yan Ma[39] used the RANS solver of CFD commercial software FLUENT to do the performance calculation of a 57000DWT bulk carrier. In the calculation, the SST k-ω turbulence model was used and the overall calculation progress was based on the SIMPLE arithmetic. The discrete equations were solved by Gauss-Seidel method and the algebraic multigrid is adopted to accelerate the solving speed. The simulation was done on the propeller with or without the PBCF so that the energy-saving effect was obtained. Deng Guo[40] used the CFD commercial software FLUENT to study the effect of the installation angle, the diameter and the change of the longitude position on the energy-saving effect. At the same time, the different turbulence models were used and drew that different model would bring about different results. Huibo Qi[41] used the CFD method to analyze the hydrodynamic performance of the propeller with or without the PBCF. The paper used Fortran programming language to establish the model according to the offset of the propeller and the PBCF. Also, Gambit was used to build the calculation model of the propeller and the PBCF. In the calculation, the RNG turbulence model and the MRF were used. It drew that the working principle of the PBCF was that it could weaken the vortex and improve the thrust of the propeller. Berger et al[42], used the CFD method to study the design stage of the optimization of the PBCF. Takafumi et al[43], used CFD to compare the difference while the PBCF was used on the ship model and the real ship.

4. New propeller

4.1 Contra-Rotating Propellers

Contra-rotating propeller consists of two normal propellers which are installed at the same axis and have the opposite sense of rotation. It is also called twin contra-rotating propellers. Compared to the normal propeller, its biggest advantage is that the rear propeller can absorb the rotation energy of the wake flow of the front propeller so that the propulsive efficiency is improved. If the two propellers match with each other perfectly, the wake flow of the rear propeller can almost not exist the circumference induced velocity. As a result, the propulsive performance is improved a lot.

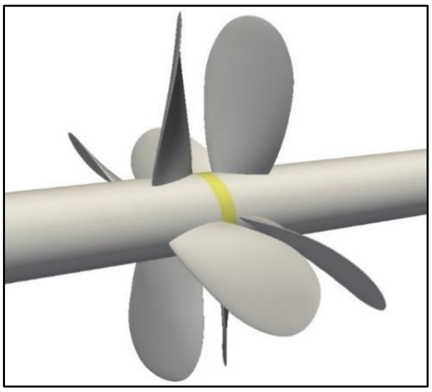
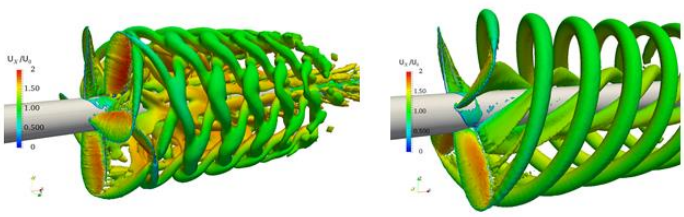



Fig. 9 Contra-Rotating Propellers

Tao Zhang[44] used the CFD commercial software FLUENT to study the effect of the unsteady of the CRPs on the hydrodynamic of the ship based on the sliding grid. In the calculation, the RHG k-ω turbulence model and the SIMPLE was used. By comparing the CFD calculation result of the unsteady slip plane model and the steady multiple reference system model, it drew that since the trailing vortex of the front propeller had the strike on the blades of the rear propeller, the unsteady slip plane model could forecast the thrust and torque of time typically averages more accurately. S. Béchet et al,[45] used several kinds of CFD software to design the CRPs with a series of stages. This paper optimized the design parameter in the process of the numerical calculation and a set of CRPs design method. Yun Li[46] used the CFD commercial software STAR-CCM+ to study the performance of the CRPs and asynchronous CRPs under different working conditions. In the calculation, the sliding grid and the VOF method which was meant to deal with the free surface were used. Linyuan Che et al,[47] used the open source software OpenFOAM and the commercial software STARCCM-CCM+ to do the unsteady numerical simulation to the hydrodynamic of the individual propeller, individual pod and the combination of the pod and the CRPs in the uniform flow. In the CFD calculation, the sliding grid was used. After the result comparing between numerical simulation and the open water test, the reliability of the CFD method was confirmed. Zhiyin Hou[48] used the CFD software FLUENT to forecast the cavitation and noise performance for a certain CRPs. Yu Chang et al,[49] used the CFD numerical simulation calculated the hydrodynamic performance. In the calculation, the mixed surface comparison method and sliding grid were used. It drew that the unsteady effect must be taken into account in the hydrodynamic numerical simulation of the CRPs. Dongya He of CMHL used the open software OpenFOAM to do the numerical simulation on the open water performance of the CRPs. The paper used sliding grid to deal with the relative rotation of the CRPs. It drew that the propulsion efficient of the CRPs was 2.2% high than the individual propeller. This result was partly due to that the rear propeller could absorb the rotation energy in the wake flow of the front propeller. This paper found that the rotating speed ratio and the blade numbers between the front and the rear propeller and the gap between the two propellers were all important factors that would affect the performance of the CRPs, fig. 9 is the comparison between the CRPS and the individual propeller, fig. 10 is the circumferential velocity distribution of the CRPs.



(a) CRPs (b) Single propeller

Fig. 10 The vortex of the CRPs and the individual propeller

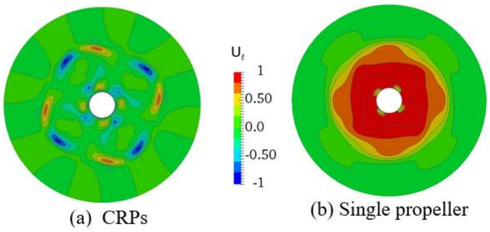


Fig. 11 the circumferential velocity distribution of the CRPs and the individual propeller

4.2 Bionic propellers

The traditional underwater vehicle is propeller propulsion which has relatively low efficient. Bionic propellers model the natural aquatic organisms which have high propulsion efficient.

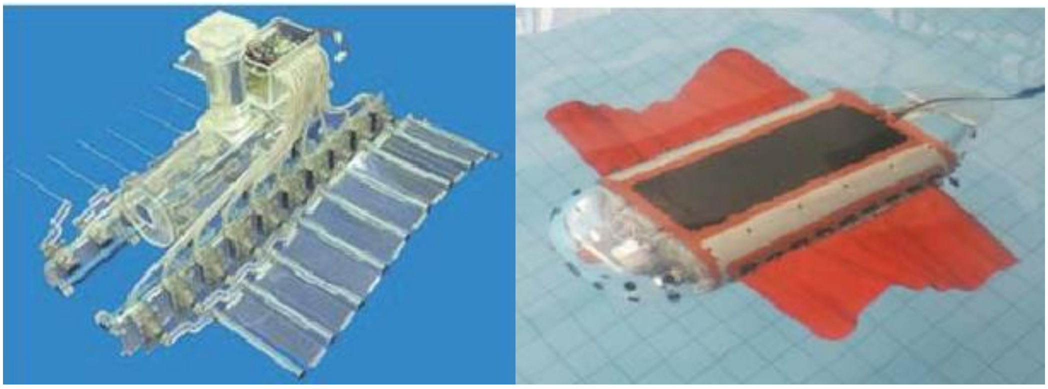


Fig. 12 Bionic undulating fins

M. Bozkurttas[50] designed a propeller imitating the fish fins and used the CFD numerical simulation to calculate the hydrodynamic performance of the new propeller. Galdo[51] used the CFD to designed a bionic undulating fins and to do the optimization of the propeller shape. Jianhui He[52] do the bionic design on the shape and structure of the skate's chest fins and used CFD to analyzed the time-variation law of the dimensionless resistance coefficient in different kinematic parameters. Han Zhou[53] used the CFD dynamic mesh to analyzed the hydrodynamic performance of the bionic undulating fins which were compared with the existing test results. Jian Li[54] focused on the sepias and designed bionic sepias propeller. The CFD was used to do the analysis on the fluid power.

5. Summary and prospect

This paper has done a summary on the CFD research of the ESDs. Based on the current research status, the future research will focus more on the following aspects.

(1) Using CFD method to optimize the ship profile and hydrodynamic ESDs at the same time;
(2) Using CFD method to research on the combination ESDs to obtain better design schemes;
(3) Doing the scale effect research on the ESDs;
(4) Researching on the bionic propulsion.

Because of the paper length, this paper has only researched on several representative ESDs. Along with the development of the CFD theory and the IT technology, the research on the ESDs using the CFD method will get a new high level.

Refearence

[1] Resolution MEPC. 203 (62), Amendments to the annex of the protocol of 1997 to amend the international convention for the prevention of pollution from ships, 1973, as modified by the protocol of 1978 relating thereto [S], 2011.

[2] Huang L, Chen L, Qiu L Y, et al. The present situation and analysis of marine hydrodynamic additional energy-saving device [J]. China Shipping, 2013(11):160-162.

[3] Jianhua Wang, Decheng Wan. Numerical simulation of pure yaw motion using dynamic overset grid technology, Chinese Journal of Hydrodynamics, 2016, Vol. 31, No. 5, pp. 567-574.

[4] Yu J W, Feng S F. The review of computational fluid dynamics [J]. Modern Manufacturing Technology and Equipment, 2013(6):25-26.

[5] Schneekluth, H., 1986. Wake equalizing duct. The Naval Architect 103: 147-150.

[6] Fahri Celik., A numerical study for effectiveness of a wake equalizing duct[J]. Ocean Engineering, 2007, 34: 2138-2145.

[7] Heinke H. J., Hellwig-Rieck K.. Investigation of Scale Effects on Ships with a Wake Equalizing Duct or with Vortex Generator Fins[C], Proceedings of 2nd International Symposium on Marine Propulsors, Hamburg, Germany, 2011.

[8] Chonghong Yin, Jianwei Wu, Decheng Wan, "A Numerical Study for Self-propelled JBC with and without Energy Saving Device", Proc. of Tokyo 2015 CFD Workshop, 2015.

[9] Martinas G, Cupsa O. Numeric Wake Equalizing Duct Geometry Optimization for a Given Ship[J]. 2015, 9(2):243-248.

[10] Huang S F, Zhao F. Study on the scale effect and energy saving mechanism of the wake equalizing duct[J]. JOURNAL OF HYDRODYNAMICS, 2016(2):171-176.

[11] Sun T, Wan D C. The numerical simulation and analysis of the wake equalizing duct[J]. JOURNAL OF HYDRODYNAMICS:, 2016(6):651-658.

[12] Sheng H L, Cai H P, Su Y M. Theoretical prediction method of energy saving effect of energy saving device in front of propeller [J]. Journal of Shanghai Jiao Tong University, 2010, 44(10):1418-1422.

[13] S Park, OH Gwangho, SH Rhee. NUMERICAL ANALYSIS OF TURBULENT FLOW AROUND ENERGY SAVING PRE-SWIRL STATOR FOR FULL AND MODEL SCALE SHIPS.

[14] Lin N J, Miao F, Huang G F. Study on the optimal design of the pre-swirl stator [J].Shipbuilding of China, 2014(4):74-81.

[15] Dang J, Dong G, Chen H. An Exploratory Study on the Working Principles of Energy Saving Devices (ESDs): PIV, CFD Investigations and ESD Design Guidelines[C]. ASME 2012, International Conference on Ocean, Offshore and Arctic Engineering. 2012.

[16] Kim K, Leer-Andersen M, Werner S, et al. Hydrodynamic optimization of pre-swirl stator by CFD and model testing[J]. International Shipbuilding Progress, 2013, 60(1):233-276.

[17] Yong-Jin Shin1, Moon-Chan Kim, Won-Joon Lee1. Numerical and Experimental Investigation of Performance of the Asymmetric Pre-Swirl Stator for Container Ship.

[18] Yan Xing-Kaeding, Scott Gatchell and Heinrich Streckwall. Towards Practical Design Optimization of Pre-Swirl Device and its Life Cycle Assessment.

[19] Wang Z Z, Xiong Y, Liu Z H, et al. The influence of the profile shape and installation angle of the shaft on the ship wake field [J]. China Ship Research, 2012, 07(4):23-29.

[20] Yan G Z, Chen L Q, Huang S Q. Evaluation and analysis of the energy-saving effect of fin

stabilizer and rudder bulb in the open water[C]. National Symposium on water dynamics. 2014.
[21] Shen H L, Obwogi E O, Su Y M. Scale effects for rudder bulb and rudder thrust fin on propulsive efficiency based on computational fluid dynamics[J]. Ocean Engineering, 2016, 117:199-209.
[22] Wang G B, Xie Y H. Calculation and analysis of the hydrodynamic performance of duct propeller with rudder ball [J]. Marine Engineering, 2013, 42(3):48-50.
[23] Li X J, Miao F, Shen H L. Numerical prediction of energy saving effect of rudder ball based on CFD Technology [J]. Ship Standardization Engineer, 2014, 47(3):18-20.
[24] Chen L Q, Huang S Q. Analysis on energy saving effect of variable parameters of rudder ball under self-propulsion [C]// National Symposium on hydrodynamics. 2015.
[25] Guo C Y, Huang S. Calculation of the hydrodynamic performance of the propeller with the rudder by the surface element method [J]. Journal of Harbin Engineering University, 2006, 27(4):501-504.
[26] Wang W Q, Wang S X, Zhang T T. Hydrodynamic performance analysis of rudder thrust fins in ship motion [J]. Ship, 2016, 27(6):19-27.
[27] Hu H Z, Su Y M, Sheng H L, et al. Study on the energy saving effect of rudder thrust fins [J]. Ship Science and Technology, 2016, 38(3):67-72.
[28] Hu W T. Study on hydrodynamic performance of rudder with rudder ball thrust fin [D]. Harbin Engineering University, 2011.
[29] Shen H L, Obwogi E O, Su Y M. Scale effects for rudder bulb and rudder thrust fin on propulsive efficiency based on computational fluid dynamics[J]. Ocean Engineering, 2016, 117:199-209.
[30] Lee S K, Yu K, Tseng K C. CFD simulation of slamming loads on a containership fitted with energy-saving rudder fins[J]. 2012.
[31] Guo C Y, Zhao Q X, Wu T C, et al. Research Progress on energy saving technology of ship additional hydrodynamic combination [J]. Ship Science and Technology, 2014, 36(4):1-10.
[32] Hou L X, Hu A K, Wang C H, et al. Research on the theory design method of Grim vane wheel after propeller [J]. Journal of Ship Mechanics, 2015(8):905-911.
[33] OUCHI K, OGURA M, KONO Y, et al. A research and development of PBCF(propeller boss cap fins)-improvement of flow from propeller boss[J]. Journal of Society of Naval Architects of Japan. 1988, 163: 66-78.
[34] OUCHI K, TAMASHIMA M, KAWASAKI T, et al. A research and development of pbcf(propeller boss cap fins)-2nd report: study on propeller slipstream and actual ship performance[J]. Journal of Society of Naval Architects of Japan. 1989,165: 43-53.
[35] TAKEO N, NORIO I, HISASHI K. Energy saving technology of pbcf(propeller boss cap fins) and its evolution[C]//IPS'10. Okayama, Japan, 2010.
[36] Ma Y, Xin G Z, Shi X Y, et al. Numerical evaluation and test of Propeller Boss Cap Fins [J]. Shipbuilding of China, 2011, 52(7): 7-18.
[37] Mizzi K, Demirel Y K, Banks C, et al. Design optimisation of Propeller Boss Cap Fins for enhanced propeller performance[J]. Applied Ocean Research, 2017, 62:210-222.
[38] Shi X Y, Ma Y. Research on the 82000DWT bulk cargo ship with propeller boss Cap Fins [C]// National Conference on Hydrodynamics. 2011.
[39] Ma Y, Xin G Z, Shi X Y, et al. Numerical evaluation and test of Propeller Boss Cap Fins [J]. Shipbuilding of China, 2011, 52(S1):7-18.

[40] Guo D, Li T L, Hu J M. Numerical simulation of propeller PBCF energy saving performance [J]. China Ship Research, 2014, 9(1):59-65.
[41] Qi F H, Liu Y B, Zhang K Z, et al. Analysis of the energy saving mechanism of PBCF [J]. Ship Science and Technology, 2016, 38(13):138-142.
[42] S Berger， M Druckenbrod， M Pergande. Testing a semi-automated tool for the optimisation of full-scale marine propellers working behind a ship. 2013.
[43] Kawamura T, Ouchi K, Nojiri T. Model and full scale CFD analysis of propeller boss cap fins (PBCF)[J]. Journal of Marine Science and Technology, 2012, 17(4):469-480.
[44] Zhang T, Yang C J, Song B W, et al. CFD simulation of unsteady performance of CRPs [J]. Journal of Ship Mechanics, 2011, 15(6):605-615.
[45] Bechet S, Negulescu C, Chapin V, et al. Integration of CFD tools in aerodynamic design of contra-rotating propellers blades[J]. 2011.
[46] Li Y. Hydrodynamic Performance Prediction of Contra-Rotating Propeller Under Different Working Conditions and The Research of Non Synchronous Counter-Rotating Propeller [D]. Harbin Engineering University, 2015.
[47] Che L Y, Du Y L, Chen W M. The performance research on the CRPs and pod[C]. Ship hydrodynamics Conference. 2015.
[48] Hou Z Y, Wang C, Bai X F. Simulation and prediction of non cavitating noise of contra- rotating propeller based on CFD [J]. Marine Engineering, 2015(4):37-40.
[49] Chang Y, Hong F W, Zhang Z R, et al. The numerical analysis on the hydrodynamic performance of the CRPs[C]// Conference on ship hydrodynamics and the ittc30 anniversary of China's shipbuilding industry. 2008.
[50] Bozkurttas M, Dong H, Mittal R, et al. CFD-Based Analysis and Design of Biomimetic Flexible Propulsor for Autonomous Underwater Vehicles[J]. Aiaa Journal, 2013.
[51] Galdo M I L, Vidal C G R, García J D R. Optimization of the efficiency of a biomimetic marine propulsor using CFD[J]. Ingeniería E Investigación, 2014, 34(1):17-21.
[52] He J H, Zhang Y H. Design and kinematics simulation analysis of bionic undulating fin propeller [J]. System Simulation Technology, 2016, 12(3):173-178.
[53] Zhou H. Computational and Experimental Hydrodynamic of Underwater Robots Propelled by Bionic Undulating Fins[D]. National University of Defense Technology, 2009.
[54] Li J. Bionic squid propulsion and research on its hydrodynamic simulation and experiment [D]. Harbin Institute of Technology, 2011.