Proceedings of the Thirtieth (2020) International Ocean and Polar Engineering Conference **www.isope.org**
Shanghai, China, October 11-16, 2020

ISBN 978-1-880653-84-5; ISSN 1098-6189

CFD Investigations of Ship Propulsion Performance at Different Trim Angles

Jianhua Wang[1], Decheng Wan[1], Moustafa Abdel-Maksoud[2]*

[1] Computational Marine Hydrodynamics Lab (CMHL), State Key Laboratory of Ocean Engineering,
School of Naval Architecture, Ocean and Civil Engineering, Shanghai Jiao Tong University, Shanghai, China
[2] Institute for Fluid Dynamics and Ship Theory, Hamburg University of Technology, Hamburg, Germany
[*]Corresponding author

ABSTRACT

In the present work, ship propulsion performance at different trim angles are investigated. CFD method with dynamic overset grid technique is used to direct simulate the free running ship model. The self-propulsion with design ship speed Fr=0.20 is firstly conducted and the predicted results are compared with experimental data. Then ship with different trim angles varying from -0.8° to 1.6° are simulated to investigate the propulsion behavior at different trim condition. All the numerical simulations are carried out by CFD solver naoe-FOAM-SJTU. Hydrodynamic forces including thrust and torque of twin propellers are analyzed. It is found that the thrust and torque have much different behavior at negative and positive trim angle. The corresponding wake flows at different trim angles are responsible for the above propulsion characteristics.

KEY WORDS: Ship propulsion; trim effect; overset grid method; computational fluid dynamics; ship hydrodynamics.

INTRODUCTION

Recently, the performance of ship propulsion, especially for the efficiency of propulsion, has become a very hot topic due to the strict terms proposed by International Maritime Organization for the newly built commercial ships to meet the increasing requirement of Energy Efficiency Design Index (EEDI) regulations. Thus, it is of great importance to find a suitable and reliable approach to evaluate the ship propulsion performance under various conditions. Generally, ship propulsion is closely related to resistance and the propeller performance. In order to evaluate a ship's propulsion performance, firstly we need to have the ability to estimate the behavior of ship hull-propeller interaction, and another important aspect is the resistance prediction at different conditions. Both the above two aspects can contribute to improve the efficiency of ship propulsion. For the hull-propeller interaction, it is very essential to have an accurate prediction model of the hydrodynamic forces of self-propelled ship to see whether the reduction of power will affect the performance of ship propulsion. For the ship resistance, ship trim optimization will have a good influence on the hydrodynamic forces of ship hull and further finding a propriate position to reduce the resistance. Therefore, trim angle effects on the ship propulsion should be draw more attention.

Regarding to the prediction of ship propulsion with ship-propeller interaction, there are mainly two approaches, one is the experimental study and another one is the computational fluid dynamic (CFD) simulations. Experimental tests in a towing tank with free-running model is widely used to predict the performance of ship propulsion. The experiments can give reliable data, while the facility cost is very expensive and the test procedure are still very complex. The disadvantage can be more obvious when we want to get the detailed flow information for further analysis. So far, many researchers are using the CFD approach to predict the ship-propeller interactions and the previous studies can be classified into two categories based on the propeller description: body-force model and discretized propeller model. The body-force model is more efficient when compared with the simulations using fully discretized propeller model. Gaggero et al. (2017) carried out self-propulsion simulation using coupled BEM/RANS approach and the authors noted that the more work need to be done to improve the accuracy of the coupled model. Gokce et al. (2018) performed CFD simulations of self-propulsion based on RANSE approach, where a virtual disk propeller model are used to compare with empirical methods. It is concluded that the body force model has a better performance than empirical method with respect to empirical relations advised by IMO.

Apart from the body force propeller model, fully discretized propeller using dynamic overset grid method has been successfully applied to the CFD simulations of ship-propeller interaction. Carrica et al. (2010) computed the self-propulsion of KCS model free to trim and sinkage with using discretized propeller and the results show good agreement. Castro et al. (2011) also simulated the self-propulsion of KCS model but at full scale using discretized propeller. Bekhit (2018) conducted both body force propeller model and fully discretized propeller model simulations of JBC ship self-propulsion. It showed that the discretized propeller simulations can give a better description of the full details of flow characteristics, while body force model can stand as a sufficient tool for quick prediction. Shen et al. (2015) implemented dynamic overset grid module to naoe-FOAM-SJTU solver and applied to the simulation of KCS self-propulsion. Wang et al. (2016) performed CFD simulation of self-propulsion for a single-screw ship in shallow water using RANS method coupled with fully discretized propeller. Further work by extending the solver to self-propulsion in waves has also been done (Wang et al., 2017).

Ship trim optimization has also been a very attractive study in the self-propulsion. Early studies mostly focus on the relations between trim angle and ship resistance (bare hull model). Sherbaz and Duan (2014) investigated ship trim angle and its influences on ship resistance for a container ship and proposed a computational technique for ship trim optimization. Islam and Guedes Soares (2019) carried out numerical studies of trim effects on the ship resistance at different ship speeds and draft. It is concluded that the optimum trim condition varies with both ship forward speed and draft condition, and the variation does not appear to follow any trend.

It can be seen that CFD simulations using fully discretized propeller model has been successfully applied to the prediction of ship self-propulsion. Ship trim effect on ship resistance has also been done for bare hull model. However, the trim effects for the free running ship model has barely been done. The objectives of this study are to investigate the trim effects on ship propulsion of a free running ship model. The experimental data of self-propulsion at ballast condition is used to validate the numerical approach. Then ship self-propulsion at different trim angle will be simulated to investigate the trim effects on the propulsion performance. The detailed flow field around propellers will also be presented to explain the hydrodynamic performance.

The paper is organized as follows, the following section will describe the numerical approach, then computational overviews including ship model, grid distribution, boundary conditions are presented. Following that, numerical results for self-propulsion at different trim angles, along with flow visualizations are discussed in detail. Finally, a brief conclusion of this paper is drawn.

NUMERICAL APPROACHES

CFD Solver

The present simulations are based on the in-house CFD solver naoe-FOAM-SJTU (Wang et al., 2019a). The abbreviation of "naoe" stands for naval architecture and ocean engineering. The present solver is developed for complex marine hydrodynamic problems. The main feature of naoe-FOAM-SJTU solver is the self-developed modules including dynamic overset grid and 6DoF motion module with a hierarchy of bodies (Shen et al., 2015), which is very convenient to perform direct simulations of free running ship incorporating with rotating propellers and turning rudders. Other modules of the solver includes a 3D numerical wave tank (Cao and Wan, 2014; Shen and Wan, 2016), a mooring system module (Liu and Wan, 2013), delayed detached eddy simulation module (Zhao et al., 2018), etc. Up to now, the CFD solver has been successfully applied to predict the hydrodynamic performance of ship resistance and wave-making (Wang et al., 2020), seakeeping, propulsion(Shen et al., 2015; Wang et al., 2019b), maneuverability (Wang and Wan, 2020) and ship maneuvering in waves (Wang and Wan, 2018; Wang et al., 2017, 2018). naoe-FOAM-SJTU solver calculates Reynold-Averaged Navier-Stokes equations for unsteady, incompressible, immiscible two-phase flows. An algebraic volume of fluid (VOF) method coupled with artificial compression technique (Berberović et al., 2009) in OpenFOAM is applied to capture the free surface.
Built-in numerical schemes in OpenFOAM are employed to discretize and solve the partial differential equations. The convection terms in momentum equations are discretized by a second-order TVD limited linear scheme, and the diffusion terms are approximated by a second-order central difference scheme. A second-order backward scheme is used for temporal discretization except for the VOF advection equation, where implicit Euler scheme is adopted. Van Leer scheme is used for the convection term in VOF equations.

Turbulence Model

One of the main features of ship flows is the high Reynold number varying from 1×10^6~1×10^9 (model scale to full scale). Thus, it is very important to choose a practical turbulence model to predict the high Re flows. In the present work, turbulence is modelled with the standard shear stress (SST) *k-ω* two-equation model (Menter et al., 2003). The parameters in turbulent kinetic energy (TKE) and dissipation rate equations are chosen by common guidelines using SST *k-ω* model.

Single-run Approach

In the present paper, different trim angles are adopted to predict the trim effects on the ship propulsion performance. In order to simulate the trim angle in a more efficient way, we are using a single-run approach to achieve different trim angles in one computation run. The procedure for single-run is to give a very small acceleration value for pitch motion and the trim angle can increase or decrease in a certain region. The pitch motion can be described as:

$$\theta = a_p t \tag{1}$$

where a_p is the acceleration rate of pitch motion and in the present simulations $a_p = 0.2 \text{ deg.}/s$ is adopted, which means that the trim angle will be increase 1 degree in 5 seconds in model scale.

COMPUTATIONAL OVERVIEWS

Geometry model

The twin-screw fully appended ONR Tumblehome ship model 5613 is used for all the self-propulsion simulations. The ship model is fitted with skeg bilge keels, shaft, brackets and rudder root. The 3D geometry model of ONR Tumblehome is shown in Fig. 1, and the main particulars are listed in Table 1. This ship model is 3.048m long and it is used as one of the benchmark ship models in Tokyo 2015 CFD Workshop and the coming SIMMAN 2020 workshop. The available experimental results can be used to validate our CFD simulations.

Table 1. Main particulars of ONR Tumblehome ship model

Main particulars		Model scale	Full scale
Length of waterline	$L_{WL}(m)$	3.147	154.0
Beam of waterline	$B_{WL}(m)$	0.384	18.78
Draft	$T\ (m)$	0.112	5.494
Displacement	$\Delta\ (kg)$	72.6	8.507e6

Propeller diameter	D_P (m)	0.1066	NA
Propeller shaft angle	ε $(deg.)$	5	NA
Propeller rotation		inward	inward

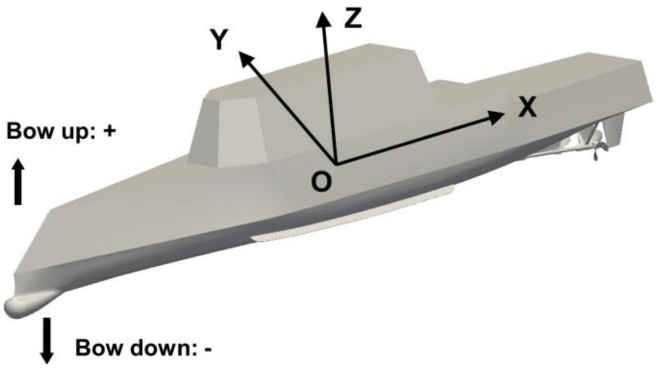


Fig. 1 Geometry of ONR Tumblehome ship model

The coordinate systems for the ship model is shown in Fig. 1. The x axis is towards the stern and positive for y axis is towards the starboard side. For the trim angles, bow up is positive and bow down is negative.

Computational Grids

Free running ship model is simulated using dynamic overset grids and in order to directly handle the large amplitude ship motions and twin rotating propellers and moving rudders, the computational domain is divided into six overlapping part: background, ship hull, two for propellers and another two for twin rudders. The distribution of computational grids and local grid arrangement around ship hull, propeller and rudder is shown in Fig. 2.

The computational domain of the simulation ship self-propulsion in each part is as follows: background domain extends to -1.5Lpp < x <4.0Lpp , -1.5Lpp < y < 1.5Lpp , -1.0Lpp < z < 0.5Lpp , and the range of hull domain is -0.15Lpp < x < 1.2Lpp , -0.13Lpp < y < 0.13Lpp , -0.25Lpp < z < 0.25Lpp . All grids used in this paper are generated by the commercial software HEXPRESS, a mesh generation tool provided by NUMECA. The total grid number of the simulation is 7.34M. It should be noted that artificial gaps between propeller and shaft, rudder and rudder root are used to obtain enough interpolation cells, where at least 8 cells have been arranged in the gap. The y+ value is around 40 along the hull surface with the consideration of wall functions are applied in the near wall region.

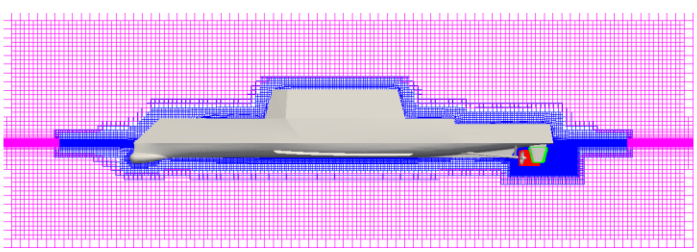


Fig. 2 Profile view of computational grids

Boundary Conditions

The computational domain and the boundary conditions are depicted in Fig. 3. Since the present simulations are in deep water, so the two lateral sides and the bottom are set as farfield boundary. Other boundaries are inlet, outlet, and atmosphere. Ship hull has a wall type boundary. The detailed boundary conditions set for each one in the present simulations are shown in Table 2.

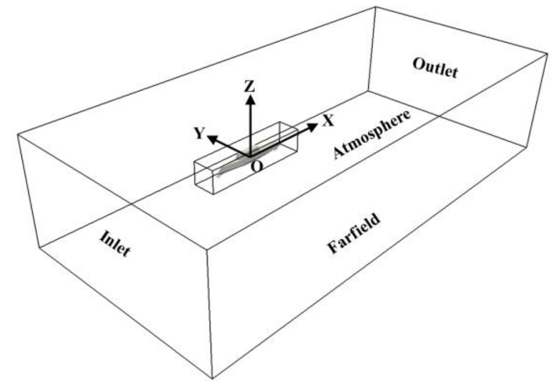


Fig. 3 Computational domain and boundary conditions

The detailed boundary conditions for each patch is shown in abbreviation format, where FV is Fixed Value, OPMV is Outlet Phase Mean Velocity (an outlet boundary condition implemented in OpenFOAM), SP is Symmetry Plane, PIOV is Pressure Inlet Outlet Velocity that imposes zero gradient for the out flow, while inflow velocity is using the normal component value, and MWV is moving Wall Velocity. For the pressure conditions, FFP is fixed Flux Pressure that adjusts the pressure gradient such that the flux on the boundary is related with the velocity boundary condition; ZG is Zero Gradient, TP stands for Total Pressure, which is calculated as the sum of static pressure reference and the dynamic component. It should also be aware that the wall functions are applied in the k, ω and eddy viscosity conditions.

Table 2. Boundary conditions in the simulations

Boundaries	U	p_rgh
Inlet	FV	FFP
Outlet	OPMV	ZG
Farfield	SP	SP
Atmosphere	PIOV	TP
Hull	MWV	FFP

NUMERICAL RESULTS AND DISCUSSION

In the present study, the ship is advancing in calm water with a target ship speed of 1.11m/s, corresponding to Fr=0.2. The self-propulsion case at original ballast condition is conducted to get the rotational speed of propeller and this is also the benchmark case in SIMMAN 2020 workshop. The calculated data will be validated with the available experiments. Numerical computations are carried out on the HPC cluster center in Computational Marine Hydrodynamics Lab (CMHL), Shanghai Jiao Tong University. Each node consists of 2 CPUs with 20 cores per node and 64GB accessible memory (Intel Xeon E5-2680v2 @2.8 GHz). 40 processors are assigned to calculate the self-propulsion case, in which 39 processors are assigned for the flow calculation and the other one processor is applied for the DCI computation using overset grids. The time step was set to $5\times10^{-4}s$, which corresponds to approximately 1.5 degrees of propeller rotation per time step. Numerical simulations for validation case and different trim angle case will be discussed in the following two sections.

Validation Case

The present simulation follows the benchmark case of self-propulsion in Tokyo 2015 CFD Workshop on ship hydrodynamics and the experimental results are also available in SIMMAN 2020 website. The initial ship speed is U=1.11m/s with corresponding Froude number of 0.20. During the simulation, the rate of revolutions of the propeller *n* is to be adjusted to obtain force equilibrium in the longitudinal direction by PI controller (Shen et al., 2015). The proportional and integral value is set to P=800 and I=800 with the consideration of larger constants can accelerate the convergence of the propeller revolution rate and reduce the total computation time. The computations start from the stable state of towing condition for bare hull, then the ship model is released in 6 degrees of freedom following the experiment's setup.

Fig. 4 illustrates the comparison between time histories of computed rotational speed of propeller and experimental measurement. The predicted RPM is 525 and is under-estimated by 2.4% compared with experimental result of 538.

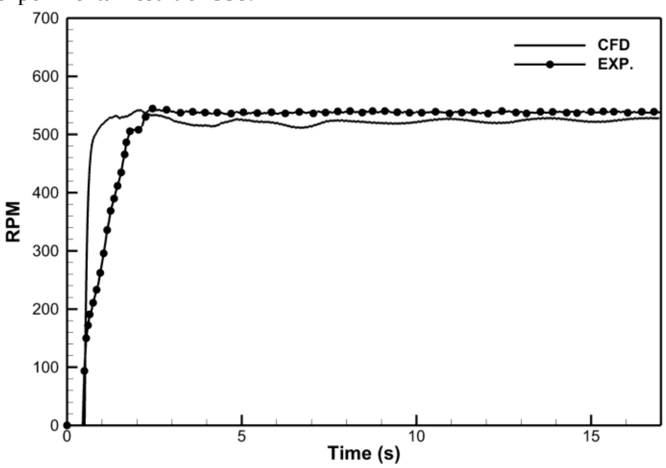


Fig. 4 Comparison of time histories of RPM between CFD prediction and experimental measurement

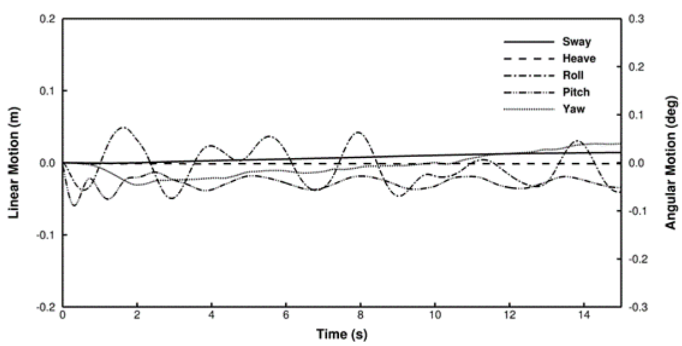


Fig. 5 Predicted ship motions for self-propulsion in calm water

Fig. 5 demonstrates the predicted 6DoF motions during self-propulsion simulation. All the ship motions are very small, where the linear motions are less than 0.1m and angular motions are less than 0.1 degrees. The comparison between predicted results and experimental data are shown in Table 3. It can be seen that good agreement is achieved and the present CFD solver can be reliable in predicting the self-propulsion performance. It can lay a very good foundation for the further numerical investigation on ship self-propulsion at different trim angles.

Table 3. Comparison of ship motions and RPM of self-propulsion in calm water

Parameters	CFD	EFD	Error
RPM	525	538	-2.4%
Sinkage ($\times 10^2$)	0.243	0.226	6.5%
Trim (deg.)	-0.0435	-0.0386	12.7%

Trim Effects on Self-propulsion

As mentioned in the numerical approach, the present paper adopted the single-run approach to predict the propulsion at different trim angles in a single computation. Trim angles varying from -0.8 degree to 1.6 degree are considered. The simulations started from the steady state of the self-propulsion at ballast condition and then the ship trim angle is under a constant acceleration to achieve different trim angles. In order to reduce the variables in the analysis, here other motions including roll, sway, yaw and heave motion are fixed.

Fig. 6 illustrates the single-run approach in the present simulations. Case A means the ship bows up during the simulation from ballast condition to trim angle of 1.6 degrees. Case B represents the ship bows down to a trim angle of 0.8 degree.

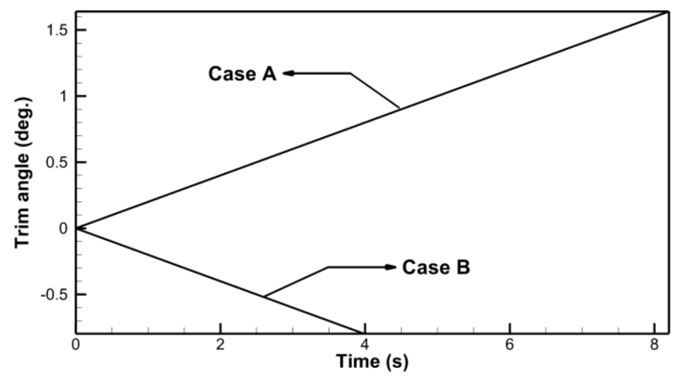


Fig. 6 Description of single-run approach

For CASE A, the numerical results of the propeller forces, including the thrust and torque are presented in Fig. 7 and Fig. 8.

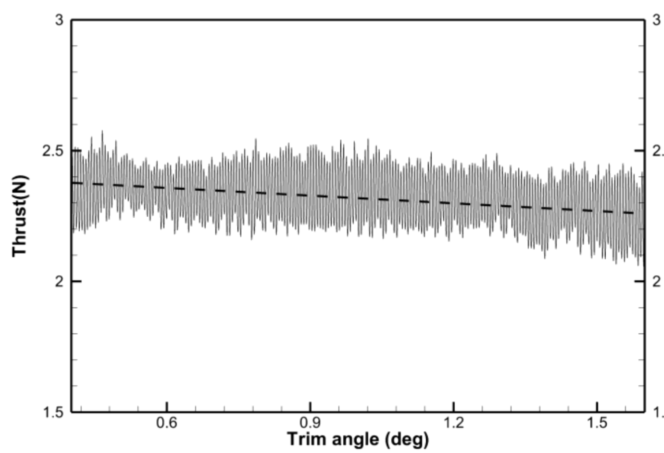


Fig. 7 Time histories of thrust at different trim angles

It is obvious that with the increasing of trim angle, both thrust and torque decrease to some extent. The high-frequency fluctuation shown in thrust and torque are due to the four rotating propeller blades passing through the flow. The dashed line in the figure is the total trend for the averaged value. The detailed information of the thrust and torque (mean value) at different trim angles are listed in Table 4.

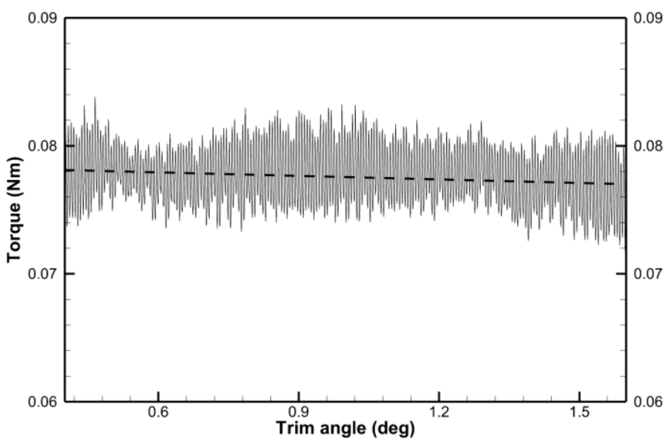
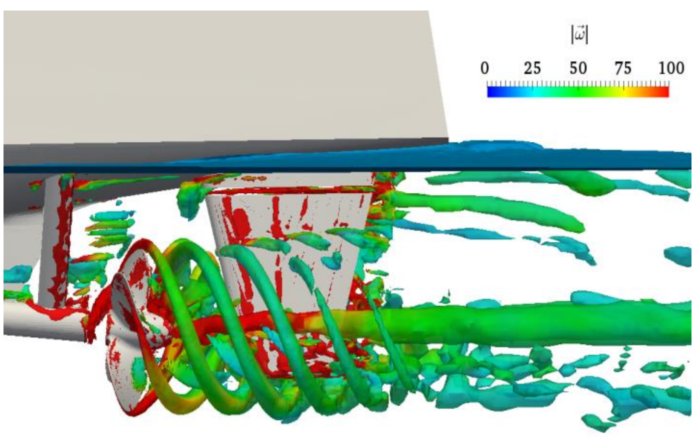


Fig. 8 Time histories of torque at different trim angles

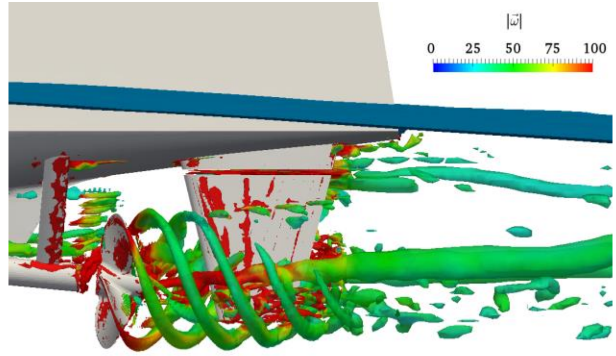
Table 4 Thrust and torque at different trim angles

Parameter	θ=-0.8°	θ=0°	θ=0.8°	θ=1.6°
Thrust (N)	2.44	2.39	2.35	2.23
Torque (Nm)	0.0782	0.078	0.0777	0.0758

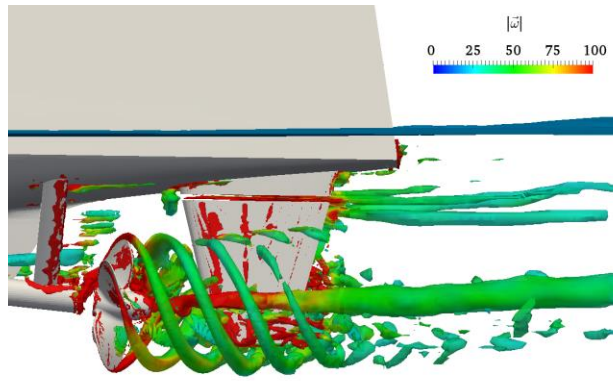
It can be clearly seen that the propulsion parameters changes with the variation of trim angles. Thrust will be increased when the ship is bowing down and the thrust at trim angle of θ=-0.8° is 2.44N, which is 8.6% larger than that of θ=1.6° condition. However, the difference of torque is not so significant in the present trim angles, where the largest deviation is 3%.



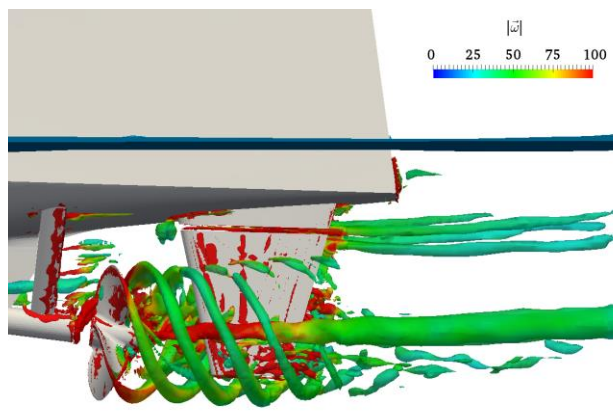
(a) θ=-0.8°



(b) θ=0°



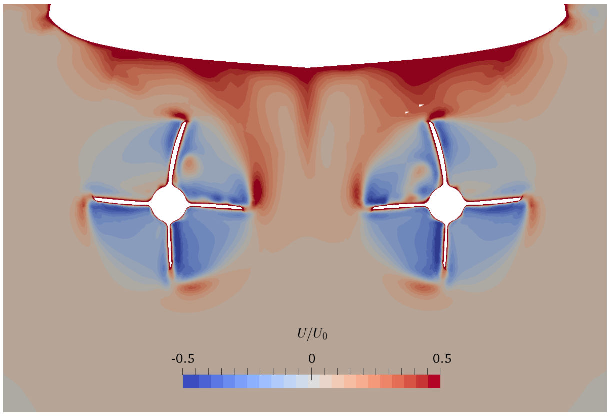
(c) θ=0.8°



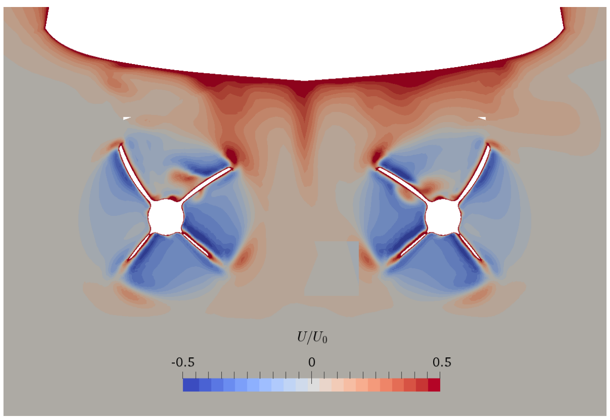
(d) θ=1.6°

Fig. 9 Vortical structures at different trim angles

In order to explain the propulsion behavior at different trim angles, detailed flow visualizations, such as 3D vortical structures around propellers and rudders, wake flow in the propeller disk, are presented. It can be noticed that the vortical structures changes significantly at different trim angles. The hub vortices separated from the propeller is very obvious and the trajectory of the hub vortex follows the ship pitch motion. In Fig. 9a ship stern is up and the rudder root is near the free surface, and the vortices are very close to the interface. In Fig. 9d the propeller is relatively far away from the free surface and the rudder root vortices and propeller hub vortices are both very obvious and the tail direction is opposite to the bowing down case.



(a) θ=-0.8°



(b) θ=1.6°

Fig. 10 Wake flow at propeller disk at different trim angles

Fig. 10 presents the wake field at propeller disk at different trim angles and the map is colored by axial velocity non-dimensioned by ship speed. It is obvious that the inflow at different trim angles can differ a lot. The inflow at θ=1.6° is smaller than that at θ=-0.8°, which means that the advance ratio J is larger when the ship bows down. This phenomenon can explain the difference of thrust and torque shown in Table 4.

CONCLUSIONS

The present paper discusses the trim effects on the self-propulsion performance of fully appended ONR Tumblehome ship model in calm water. Numerical simulations are carried out by CFD solver naoe-FOAM-SJTU. During the process, a single-run approach is adopted to achieve the simulation of different trim angles. The self-propulsion at ballast condition is firstly conducted to validate our numerical methods. The CFD solver with overset grid technique is proved to be reliable in predicting the self-propulsion performance by comparing with the available experiment. The trim effects on the self-propulsion is discussed and it is found that the thrust and torque decreases when the ship bows up. The difference can be 8.6% between the trim angle of θ=-0.8° and θ=1.6°. Another phenomenon is that the difference of torque is not as large as the thrust. The changing of thrust has been explained by the wake flow at propeller disk. In addition, the flow visualizations including 3D vortical structures around ship hull, propeller and rudder are also presented to demonstrate the flow behaviors of self-propulsion at different trim angles.

The present work didn't consider the ship motions when changing the trim angle and it will affect the accuracy of predicted results. In the future we will conduct simulations for different loading conditions separately to give more accurate results.

ACKNOWLEDGEMENTS

This work is supported by the National Natural Science Foundation of China (51809169, 51879159), The National Key Research and Development Program of China (2019YFB1704200, 2019YFC0312400), Chang Jiang Scholars Program (T2014099), Shanghai Excellent Academic Leaders Program (17XD1402300), and Innovative Special Project of Numerical Tank of Ministry of Industry and Information Technology of China (2016-23/09), to which the authors are most grateful.

REFERENCES

Bekhit, AS (2018). "Numerical simulation of the ship self-propulsion prediction using body force method and fully discretized propeller model," *IOP Conf Ser Mater Sci Eng*, 400, 042004.

Berberović, E, van Hinsberg, N, Jakirlić, S, Roisman, I, and Tropea, C (2009). "Drop impact onto a liquid layer of finite thickness: Dynamics of the cavity evolution," *Phys Rev E*, 79(3), 36306.

Cao, H, and Wan, DC (2014). "Development of Multidirectional Nonlinear Numerical Wave Tank by naoe-FOAM-SJTU Solver," *Int J Ocean Syst Eng*, 4(1), 52–59.

Carrica, PM, Castro, AM, and Stern, F (2010). "Self-propulsion computations using a speed controller and a discretized propeller with dynamic overset grids," *J Mar Sci Technol*, 15(4), 316–330.

Castro, AM, Carrica, PM, and Stern, F (2011). "Full scale self-propulsion computations using discretized propeller for the KRISO container ship KCS," *Comput Fluids*, 51(1), 35–47.

Gaggero, S, Villa, D, and Viviani, M (2017). "An extensive analysis of numerical ship self-propulsion prediction via a coupled BEM/RANS approach," *Appl Ocean Res*, 66, 55–78.

Gokce, MK, Kinaci, OK, and Alkan, AD (2018). "Self-propulsion estimations for a bulk carrier," *Ships Offshore Struct*, 0(0), 1–8.

Islam, H, and Guedes Soares, C (2019). "Effect of trim on container ship resistance at different ship speeds and drafts," *Ocean Eng*, 183, 106–115.

Liu, Y, and Wan, DC (2013). "Numerical simulation of motion response of an offshore observation platform in waves," *J Mar Sci Appl*, 12(1), 89–97.

Menter, FR, Kuntz, M, and Langtry, R (2003). "Ten years of industrial experience with the SST turbulence model," *Turbul Heat Mass Transf*, 4(1), 625–632.

Shen, Z, and Wan, DC (2016). "An irregular wave generating approach based on naoe-FOAM-SJTU solver," *China Ocean Eng*, 30, 177–192.

Shen, Z, Wan, DC, and Carrica, PM (2015). "Dynamic overset grids in OpenFOAM with application to KCS self-propulsion and maneuvering," *Ocean Eng*, 108, 287–306.

Sherbaz, S, and Duan, W (2014). "Ship trim optimization: assessment of influence of trim on resistance of MOERI container ship," *Sci World J*, 2014, 603695: 1-6.

Wang, J, and Wan, DC (2018). "CFD Investigations of Ship Maneuvering in Waves Using naoe-FOAM-SJTU Solver," *J Mar Sci Appl*, 17(3), 443–458.

Wang, J, and Wan, DC (2020). "CFD study of ship stopping maneuver

by overset grid technique," *Ocean Eng*, 197, 106895.
Wang, J, Liu, X, Wan, DC, and Chen, G (2016). "Numerical Prediction of KCS Self-Propulsion in Shallow Water," *Proc 26th Int Ocean Polar Eng Conf*, Rhodes, ISOPE, 757–763.
Wang, J, Zou, L, and Wan, DC (2017). "CFD simulations of free running ship under course keeping control," *Ocean Eng*, 141, 450–464.
Wang, J, Zou, L, and Wan, DC (2018). "Numerical simulations of zigzag maneuver of free running ship in waves by RANS-Overset grid method," *Ocean Eng*, 162, 55–79.
Wang, J, Zhao, W, and Wan, DC (2019a). "Development of naoe-FOAM-SJTU solver based on OpenFOAM for marine hydrodynamics," *J Hydrodyn*, 31(1), 1–20.
Wang, J, Zhao, W, and Wan, DC (2019b). "Simulations of Self-Propelled Fully Appended Ship Model at Different Speeds," *Int J Comput Methods*, 16(3), 1840015-1–22.
Wang, J, Ren, Z, and Wan, DC (2020). "Study of a Container Ship with Breaking Waves at High Froude Number Using URANS and DDES Methods," *J Ship Res*, https://doi.org/10.5957/JOSR.09180081.
Zhao, W, Zou, L, Wan, DC, and Hu, Z (2018). "Numerical investigation of vortex-induced motions of a paired-column semi-submersible in currents," *Ocean Eng*, 164, 272–283.