J. Marine Sci. Appl. (2010) 9: 262-267
DOI: 10.1007/s11804-010-1005-6

Applying Periodic Boundary Conditions to Predict Open Water Propeller Performance

Chao Wang*, Sheng Huang, Xin Chang and Miao He

College of Shipbuilding Engineering, Harbin Engineering University, Harbin 150001, China

Abstract: Mathematical models of propellers were created that investigate the influence of periodic boundary conditions on predictions of a propeller's performance. Thrust and torque coefficients corresponding to different advance coefficients of DTMB 4119, 4382, and 4384 propellers were calculated. The pressure coefficient distribution of the DTMB 4119 propeller at different sections was also physically tested. Comparisons indicated good agreement between the results of experiments and the simulation. It showed that the periodic boundary condition can be used to rationally predict the open water performance of a propeller. By analyzing the three established modes for the computation, it was shown that using the spline curve method to divide the grids can meet the calculation's demands for precision better than using the rake cutting method.
Keywords: propeller; open water performance; periodic boundary condition; pressure coefficient
Article ID: 1671-9433(2010)03-0262-06

1 Introduction

With the rapid development of computational fluid dynamics (CFD) technology, CFD software is also applied extensively in predicting propeller hydrodynamic performance. A big enough flow area should be established for calculation because of propeller working in crossing flow of exterior. If the whole propeller and its flow area need to be calculated, the number of meshes will become a big restricting factor for calculation speed and precision (Wang, 2004). Due to the uniform and periodic feature of the propeller in open water flow, adopting periodic boundary condition to calculate the flow field of single blade's channel can decrease enormously the calculation (Li, 2002; Takayuki, 2003).

This paper presents the numerical simulation of the flow region about single propeller blade's channel in open water using CFD software. The hydrodynamic coefficient curves of different blade numbers and skew angles were obtained. The feasibility of using periodic boundary condition to calculate the open water performance of propeller was proved by contrasting the open water performance of some model propeller and the pressure coefficient distribution at different sections, which were obtained respectively by the single channel method and the whole channel method.

2 Control equations and turbulence models

2.1 Control equations

The propeller working steadily in open water was taken in relative coordinate system. RANS equations are universally adaptive control equations of kinematics and kinetics in viscous fluid. So the RANS equations are also called basic equations for solving hydrodynamic performance of propeller, which can be written as follows (Wang, 2004):

$$\rho\frac{\partial(u_i u_j)}{\partial x_j}+\rho\frac{\partial(u_i)}{\partial t}=-\frac{\partial p}{\partial x_i}+\frac{\partial}{\partial x_i}[\mu_0(\frac{\partial u_i}{\partial x_j}+\frac{\partial u_j}{\partial x_i})-\frac{2}{3}\mu_0\frac{\partial u_i}{\partial x_i}\delta_{ij}-\rho\overline{u_i^{'}u_j^{'}}]+\rho f_i \tag{1}$$

where u_i and u_j are time-average speed components $(i,j=1,2,3)$, P is time-average pressure, ρ represents the density of water, μ_0 is viscosity coefficient of water, f_i is mass force and $-\rho\overline{u_i^{'}u_j^{'}}$ is Reynolds stress.

2.2 Turbulence models

Presently, there are few references in aspect of numerical calculations of propeller in the viscous flow field (Gong *et al*., 2007; Zhu *et al*., 2007; Wang *et al*., 2008; Liu and Xiong, 2007). The introduction of turbulence models for performance calculation of propeller is also deficient. Reynolds stress model (RSM) which can simulate questions more accurately completely forsakes the hypothesis of vortex viscosity and solves differential transport equations of Reynolds stress. Moreover, the impact of wall on the distribution of Reynolds stress is also considered. So RSM method was chosen to simulate the propeller performance.

RSM equation can be written as follows:

$$\frac{\partial}{\partial t}(\rho\overline{u_i u_j})+\frac{\partial}{\partial x_k}(\rho U_k\overline{u_i u_j})=D_{ij}+P_{ij}+\phi_{ij}-\varepsilon_{ij} \tag{2}$$

where D_{ij}, ϕ_{ij}, P_{ij} and ε_{ij} respectively represent diffusion term, source term, tension and strain term and

Received date: 2009-10-27.
Foundation item: Supported by the National Natural Science Foundation of China under Grant No. 10702016.
***Corresponding author Email:** wangchao0104@hrbeu.edu.cn

dissipation term. All the terms are respectively determined by the following equations:

$$D_{ij}=\frac{\partial}{\partial x_k}(\frac{\mu_t}{\sigma_k}\frac{\partial \overline{u_i^{'}u_j^{'}}}{\partial x_k})$$

$$P_{ij}=-(\overline{u_i^{'}u_k^{'}}\frac{\partial U_j}{\partial x_k}+\overline{u_j^{'}u_k^{'}}\frac{\partial U_i}{\partial x_k})$$

$$\Phi_{ij}=-C_1\varepsilon\alpha_{ij}+C_2\varepsilon(\alpha_{ik}\alpha_{kj}-\frac{1}{3}\alpha_{kl}\alpha_{kl}\delta_{ij})+$$
$$C_3kS_{ij}+C_4k(\alpha_{ik}S_{jk}+\alpha_{jk}S_{ik}-\frac{2}{3}\alpha_{kl}\alpha_{kl}\delta_{ij})+$$
$$C_5(\alpha_{ik}W_{jk}+\alpha_{jk}W_{ik})$$

Pressure strain item contains the quadratic term of anisotropic tensor of Reynolds stress, where $C_1=3.4+1.8P_{kk}/\varepsilon$; $C_2=4.2$; $C_3=\frac{4}{5}-1.3\Phi_\alpha^{1/2}$; $C_4=1.35$; $C_5=0.4$; $\alpha_{ij}=\frac{\overline{u_i^{'}u_j^{'}}}{2k}-\frac{1}{3}\delta_{ij}$; $\Phi_\alpha=\alpha_{ij}\alpha_{ij}$; $S_{ij}=\frac{1}{2}(\frac{\partial U_j}{\partial x_i}+\frac{\partial U_i}{\partial x_j})$; $W_{ij}=\frac{1}{2}(\frac{\partial U_i}{\partial x_j}-\frac{\partial U_j}{\partial x_i})$.

In this model, the coefficient of tension and strain term depends on change of Reynolds stress and generation of turbulence energy which have close correlation with wall effect. So the RSM embodies the impact of wall on the distribution of Reynolds stress.

Dissipation process mainly occurs in small scale eddy region. For a long time, with high Reynolds number, small scale eddy structure is said to tend to isotropy and anisotropic dissipation can be neglected. That is to say, the dissipation of turbulence shear stress tends to zero, while viscosity just arouses turbulence plus stress, which is called turbulence dissipation. Thus dissipation tensor ε can be simplified to the scalar form written as:

$$\varepsilon_{ij}=\frac{2}{3}\rho\varepsilon\delta_{ij} \tag{3}$$

Now the most extensively adoptive ε model is written as:

$$\frac{\partial\rho\varepsilon}{\partial t}+\frac{\partial}{\partial x_k}(\rho U_k\varepsilon)=\frac{\partial}{\partial x_k}(\frac{\mu_t}{\sigma_\varepsilon}\frac{\partial\varepsilon}{\partial x_k})-$$
$$C_{\varepsilon1}\frac{k}{\varepsilon}\rho\overline{u_i^{'}u_j^{'}}\times\frac{\partial U_i}{\partial x_j}-C_{\varepsilon2}\rho\frac{\varepsilon^2}{k} \tag{4}$$

In Eq.(4), there are diffusion term, source term and dissipation term on the right. The coefficients are given as $C_{\varepsilon1}=1.44,\ C_{\varepsilon2}=1.92,\ C_{\varepsilon3}=-1,\ C_\varepsilon=1.33$.

3 Foundation of calculation model

3.1 Foundation of geometry model

The calculation of the open water performance of propeller for different blades and skew angles has been made and compared with relevant model's experimental result. Some propellers of DTMB series are chosen as the calculated models and their geometry parameters are listed in Table 1.

Table 1 Dimensions of DTMB propeller

Model	No. of blades	Exp. area ratio A_E/A_0	Boss ratio	Skew angle/(°)	Trim	Designed advance coefficient
4119	3	0.6	0.2	0	without	0.833
4382	5	0.725	0.2	36	with	0.889
4384	5	0.725	0.2	108	with	0.889

The Cartesian coordinate system *O-XYZ* is set up in the paper: *X* coordinate is the direction of inlet flow; *Y* coordinate is the same as the blade's reference line; *Z* coordinate submits to the right-handed rule.

The surfaces of hub and cylinder with a diameter 5 times of propeller are respectively set as the inner and outer boundary surfaces of computational domain of single blade channel; the slice faces are set as periodic boundary conditions. During the process of modeling, according to change of skew angles of propeller, there are different methods to slice the blades. Details are shown as follows:

1) Directly cutting method. For example, the 4119 propeller has 3 blades and no skew. When dividing 1/3 of the propeller and the hub, directly cutting method can be used. That is to say, directly using 1/3 of the cylinder can finish the region division, as shown in Fig.1(a).

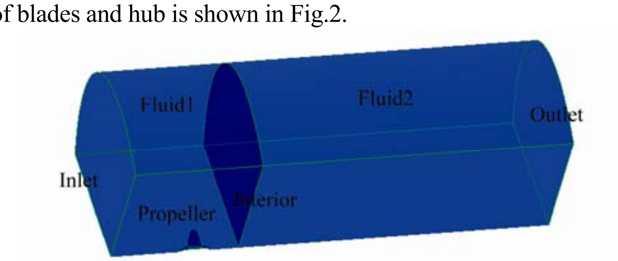
2) Rake cutting method. For example, the 4382 propeller has skew and 5 blades. The region can be divided into 3 parts, two of which are located in the front and the rear of the hub can be divided by directly cutting method. The front 2/3 part should be divided by rake cutting method because using directly cutting method will result in un-integrity of the blade. In detail, according to the skew of the blade, the computational domain should be rotated in some angle, which is shown in Fig.1(b).

3) Spline curve method. For example, the 4384 propeller is highly skewed and has 5 blades. The projections of blades on the plane have superposition. Some of the next blades will be included in the computational domain when using rake cutting method. Here it needs to create the spline curve to establish the computational domain where there is only a single blade according to the blade shape. The computational domain is shown in Fig.1(c). Choosing the spline curve is the key of modeling, which decides whether the computational domain of single blade meets the demand or not.

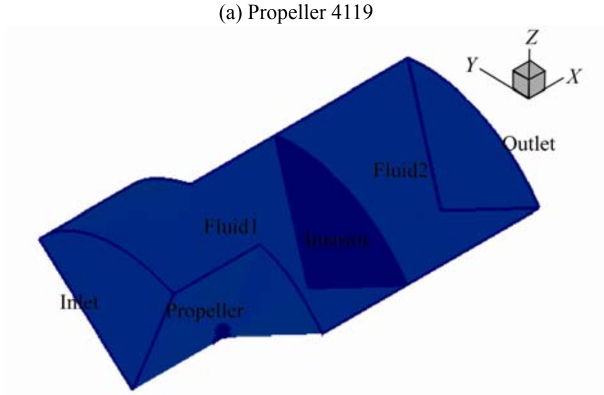
The way to choose the slice fashion can be decided by the following method. The propeller is divided into *Z* parts, if every blade is integrated, the directly cutting method can be chosen. If every blade is divided into two parts, the rake cutting method is a good slice fashion. The spline curve method should be adopted to divide the other types of propellers.

3.2 Grid division

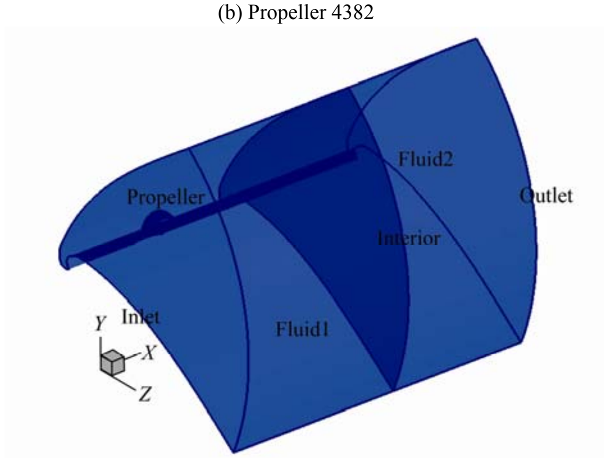
Grid is not only the geometric form of numerical model but also the carrier of simulation and analysis. Grid quality has great influence on calculation precision and efficiency. Grid division which is directly the key factor of impact on precision and efficiency takes fairly much time during CFD simulation. During grid division, local denseness method is used in the paper. In order to obtain information of the important flow region, both the tip of propeller and the joint of blade and hub are divided into denser grids. For fluid 2 which is the outlet of flow region, the density of grid division should be decreased to control the number of grids. In this way, when the number of nodes is constant, the calculation precision can be improved and the calculation amount of the smoothly changing flow field can be decreased. Grid division of blades and hub is shown in Fig.2.



(a) Propeller 4119



(b) Propeller 4382



(c) Propeller 4384

Fig.1 Computational domain of propellers

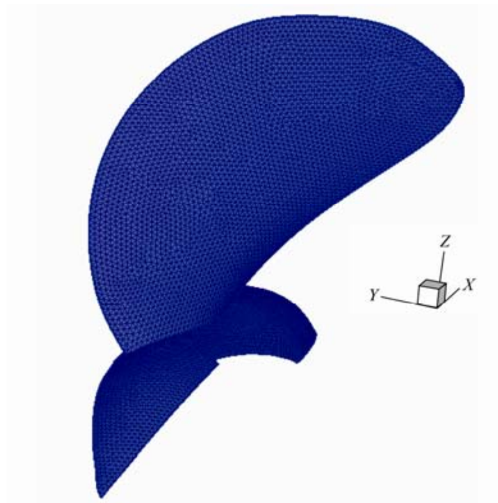


Fig.2 Grid division of blade and hub

3.3 Setting of the boundary conditions

During the calculation of open water performance of propeller, the whole computational domain rotates round some reference coordinate system and the parts can not be disturbed each other around the propeller, so more reference fluid (namely MRF model) was adopted for calculation.

Velocity inlet is set at the inlet boundary and pressure outlet is set at outlet boundary; slices of both sides of flow domain are defined as periodic boundary conditions; the interface of fluid1 and fluid2 is the interior boundary; the outer surface of big domain is set as the wall. The wall has no slip condition. Standard wall functions are adopted in near-wall treatment and influence of the wall coarseness is also taken into account.

4 Results analysis

4.1 Numerical results of open water performance

The advance velocity coefficient J of the 4119 propeller is given respectively as 0.5, 0.7, 0.833, 0.9 and 1.1. At some time, J of 4382 propeller and 4384 propeller are both given as 0.3, 0.5, 0.7 and 0.9. n =600 r/min, where n is the rotational speed of propeller, which is defined as a constant. The thrust coefficient K_T and torque coefficient K_Q in different advance velocity are solved.

By contrast of Table 2, Table 3 and Table 4, it shows that the numerical results of K_T agree with the experimental data consistently (Wang and Dong, 2005), but K_Q have a little bigger deviation. And numerical results of 4119 and 4384 propellers agree better with experimental data than 4382 propeller due to different division of computational domain. In order to use periodic boundary condition with more precision, the hub of both sides of propeller should be made the same as much as possible during division of the computational domain. The skew angle of 4382 propeller is

36 degrees and the division should be done by spline curve. In order to represent the rake cutting method, the hub should be distributed in the direction of skew as much as possible, which is shown in Fig.2. There was a bigger deviation of torque calculation by using rake cutting method. Replaced by the division method of spline curve, the deviation will be decreased.

Table 2 The comparison of the open water performance of 4119

Term	Advance velocity coefficient	0.5	0.7	0.83	0.9	1.1
K_T	Calculation data	0.29	0.207	0.1476	0.121	0.0307
	Experimental data	0.285	0.20	0.146	0.12	0.034
	Error/%	−1.79	−3.5	−1.06	−0.69	9.70
$10K_Q$	Calculation data	0.489	0.371	0.289	0.246	0.097
	Experimental data	0.477	0.36	0.28	0.239	0.106
	Error/%	−2.5	−3.06	−3.21	−2.1	8.49

Table 3 The comparison of the open water performance of 4382

Term	Advance velocity coefficient	0.3	0.5	0.7	0.9
K_T	Calculation data	0.465	0.394	0.302	0.211
	Experimental data	0.43	0.38	0.28	0.19
	Error/%	−8.14	−3.68	−7.86	−11.6
$10K_Q$	Calculation data	0.802	0.687	0.586	0.445
	Experimental data	0.87	0.75	0.66	0.51
	Error/%	7.82	8.4	11.2	12.7

Table 4 The comparison of the open water performance of 4384

Term	Advance velocity coefficient	0.3	0.5	0.7	0.9
K_T	Calculation data	0.45	0.37	0.29	0.21
	Experimental data	0.44	0.36	0.28	0.2
	Error/%	−2.27	−2.7	−3.5	−5.0
$10K_Q$	Calculation data	0.74	0.64	0.53	0.44
	Experimental data	0.71	0.66	0.5	0.46
	Error/%	−4.2	3.48	−6.0	4.35

4.2 Feasibility validation of periodic boundary condition

In order to analyze the feasibility of periodic boundary condition applied in the calculation of open water performance of propeller, 4119 propeller was illustrated in the following. The numerical results calculated by using single channel method were compared with the whole channel method. Also pressure coefficients at the radii of 0.3R, 0.7R and 0.9R were compared. $C_P = (P - P_0)/(1/2\rho V_R^2)$, where C_P is the pressure coefficient, $(P - P_0)$ represents the relative pressure. $V_R = \sqrt{V_a^2 + (2\pi nr)^2}$, where V_R is the relative advance velocity.

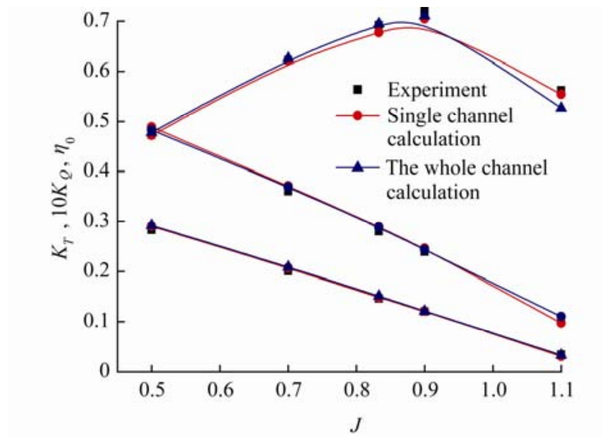
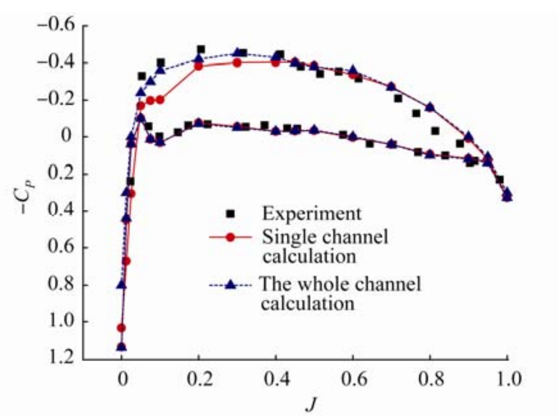
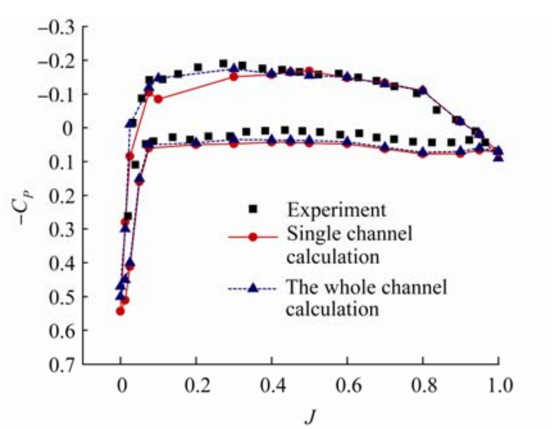


Fig.3 The performance of propeller 4 119

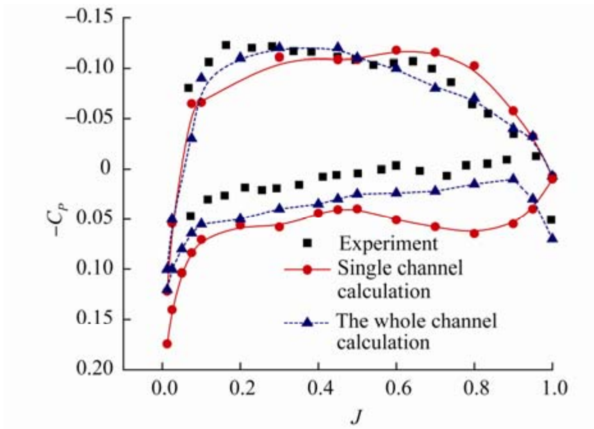
Numerical and experimental results of single channel method and the whole channel method are both shown in Fig.3. The numerical results indicated well agreement with experimental results. Just when J is bigger than 0.833, the deviation of calculation increases. Therefore using periodic boundary condition to calculate open water performance of propeller is feasible and also can save more time and improve calculation efficiency compared with the whole channel method.



(a) r=0.3R



(b) r=0.7R



(c) r=0.9R

Fig.4 Results of pressure coefficient at different sections

The pressure coefficient curves obtained by different methods are shown in Fig.4. As a whole, from the comparison of numerical results with experimental data (Su and Huang, 2003), the pressure face agreed better than the suction face and pressure coefficient C_P at the radius of 0.7R agreed best with the experimental data. At the radius of 0.3R it took the second place, mainly caused by its close to the hub at the radius of 0.3R and the hub shape has some influence on the pressure distribution. The most deviation is shown at the radius of 0.9R. The blade tip is located at the radius of 0.9R, where tip vortex often leaks and viscosity influence and fluid grads are big, so numerical results of the blade tip has some deviation. The deviation of the leading edge where x/c=0 and the trailing edge where x/c=1.0 are bigger than the central blade surface due to greater fluid grads and distinctness of pressure changing in the leading edge and the trailing edge.

From contrast of the single channel method and the whole channel method, numerical results obtained by the whole channel method are closer to the experimental data. The main deviation takes place in the leading edge and following edge, which is mostly caused by using the periodic boundary condition which neglects disturbance between blades. This is the main deficiency of using the single channel method. As a whole, it is feasible to use periodic boundary condition to calculate open water performance of the propeller because of its advantage.

5 Conclusions

Combined with RSM turbulence model, the calculation of open water performance of propellers in different skew angles with periodic boundary condition has been introduced in the paper. By contrasting calculation and experiment results, some conclusions are drawn as follows:

1) Adopting periodic boundary condition not only reduces the calculation amount and saves time but also can predict the open water performance of propeller more accurately. But as the periodic boundary condition can only be used in open water propeller of uniform flow, its application will be restricted greatly.

2) For propellers of different skew angles, the grid division of computational domain on single blade should be made by different methods and adjusted according to the actual state. And deviation of torque calculation of skew propeller obtained by Rake Cutting Method is bigger; on the contrary using spline curve to divide grids can meet the calculation precision well.

3) By contrasting with open water performance of 4119 propeller calculated by the single channel method and the whole channel method, the numerical results indicate good agreement with the experimental results. So using periodic boundary condition to calculate open water performance of propeller is feasible and also can save more time and improve calculation efficiency compared with the whole channel method.

4) Pressure coefficient curves calculated by two methods indicate the best agreement with experimental results at the radius of 0.7R and next at the radius of 0.3R. The most deviation is shown at the radius of 0.9R. The deviations in leading side and trailing side are bigger than that in the center blade. And the agreement between numerical and experimental results of the pressure face is better than the suction face.

References

Gong Lv, Yao Zheng, Wu Man (2007). Numerical analysis for hydraulic characteristics of marine propeller under reverse bent and twist. *Journal of University of Shanghai for Science and Technology*, **29**(1), 65-69.

Huang Sheng, Zhu Xiangyuang, Guo Chunyu (2007). CFD simulation of propeller and rudder performance when using additional thrust fins. *Journal of Marine Science and Application*, **6**(4), 27-31.

Huang Sheng, Wang Chao, Wang Shiyang (2009). Application and comparison of different turbulent models with computations of the propeller's hydrodynamic performance. *Journal of Harbin Engineering University*, **30**(5), 481-485. (in Chinese)

Li Daqing (2002). Validation of RANS predictions of open water performance of a highly skewed propeller with experiments. *Conference of Global Chinese Scholars on Hydrodynamics*, SSPA, Sweden, 520-528.

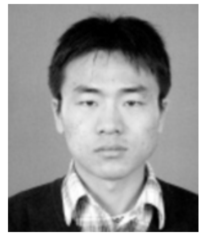
Liu Zhihua, Xiong Ying (2007). Study on the prediction of propeller open-water performance using RANS formula and multi-block hybrid meshes. *Journal of Hydrodynamics*, Ser.A, **22**(4), 450-456.

Su Yumin, Huang Sheng (2003). *The theory of ship propeller*. Harbin Engineering University Press, Harbin.

Takayuki W (2003). Simulation of steady and unsteady cavitation on a marine propeller using a RANS CFD code. *Fifth International Symposium on Cavitation,* Osaka, Japan, 1-8.

Wang Fujun (2004). *The analysis of computation fluid dynamics-the based theory and application of CFD*. Tsinghua University Press, Beijing.

Wang Guoqiang, Dong Shitang (2005). *The theory and application of ship propeller*. Harbin Engineering University Press, Harbin.
Wang Chao, Huang Sheng, Xie Xueshen (2008). Prediction of the hydrodynamic performance of propeller by CFD. *Journal of Naval University of Engineering*, **20**(4), 107-112.



Chao Wang was born in 1982. He is a doctor of Design and Construction of Naval Architecture and Ocean Structure at Harbin Engineering University. His current research interests include marine special propulsor.

Sheng Huang was born in 1945. He is a professor at Harbin Engineering University. His current research interests include shipbuilding engineering, marine propulsor and energy-saving.