J. Marine. Sci. Appl. (2010) 9: 187-191
DOI: 10.1007/s11804-010-9080-2

Discussion of Grid Generation for Catamaran Resistance Calculation

Rui Deng*, De-bo Huang, Jia Li, Xuan-kai Cheng, and Lei Yu

Multihull Ship Technology, Key Laboratory of Fundamental Science for National Defence, Harbin Engineering University, Harbin 150001, China

Abstract: In order to get some useful parameters for grid generation of catamaran, the CFD software FLUENT is used to investigate the main effects of grid generation on flow field calculation. The influences of some elements are investigated with a series of calculations in the present paper, and some alteratives are proposed. The proposed alteratives based on the analysis of the effects are used for a catamaran resistance calculation, comparisons of the calculated results with experimental data show good agreement. It shows that the research result of this paper is useful for the numerical calculation of catamaran.
Keywords: catamaran; resistance; calculation; grid generation
Article ID: 1671-9433(2010)02-0187-05

1 Introduction

CFD technology has a great development in multihull ship resistance calculation (Zou, Luo *et al.*, 2005; Chen, *et al.*, 2006; Chen, *et al.*, 2008; Deng and Huang, 2007) in recent years, but it is not considered as a fully developed technology due to the uncertain aspects such as turbulence model, simplification of the physical flow, grid generation method. All these may bring uncertainty into the numerical simulation result.

The difference between numerical simulation result and the real physical problem is called numerical error, and it is classified into three types (Tao, 2001): modeling error, discretization error and computation error. Most of the time, computation error is a small part in the numerical error. When a discretization scheme is chosen, discretization error is mainly determined by grid generation (Zhao *et al.,* 2004). Though a lot of researchers worked so much on grid generation, especially on verification and validation, no detailed result about the uncertainty of grids is gotten. Some organizations like ITTC gave a simple suggestion on the research of grids uncertainty, but the suggestion is still under discussion. The research about the uncertainty of the grids is an important part of numerical simulation, and has a direct relationship with the precision of the result.

2 CFD verification and validation

The research about the uncertainty in the numerical simulation includes verification and validation. Verification is the process that evaluates whether the equations are solved in the right way, and validation evaluates whether the right equations are solved (Zhu *et al.*, 2007).

Denoting T and S as true result and numerical result respectively, $\delta_S = S - T$ is then the difference between them, and since δ_S is composed of modeling error δ_{SM} and numerical error δ_{SN}, we have

$$\delta_S = S - T = \delta_{SM} + \delta_{SN} \tag{1}$$

$$\delta_{SN} = \delta_{SN}^* + \varepsilon_{SN} \tag{2}$$

where δ_{SN}^* is the evaluated result and ε_{SN} is the evaluated error.

$$S_C = S - \delta_{SN}^* \tag{3}$$

The uncertainty of numerical simulation U_{SN}, δ_{SN}^* and the uncertainty of error evaluating U_{S_CN} can be defined as follows:

$$\begin{cases} U_{SN}^2 = U_I^2 + U_G^2 + U_T^2 + U_P^2 \\ \delta_{SN}^* = \delta_I^* + \delta_G^* + \delta_T^* + \delta_P^* \\ U_{S_CN}^2 = U_{I_C}^2 + U_{G_C}^2 + U_{T_C}^2 + U_{P_C}^2 \end{cases} \tag{4}$$

The process of evaluating U_{SN} is that of verification, and the one during which modeling uncertainty U_{SM} is gotten is validation. If the numerical result and experimental data are denoted as S and D respectively, the error of experimental data is δ_D, and the comparison error E, the validation uncertainty U_V can be defined as follows:

$$\begin{cases} E = D - S = \delta_D - \left(\delta_{SM} + \delta_{SN}\right) \\ U_V^2 = U_D^2 + U_{SN}^2 \end{cases} \tag{5}$$

Received date: 2009-11-1.
Foundation item: Supported by the Foundation of Multihull Ship Technology, Key Laboratory of Fundamental Science for National Defence under Grant No.002010260737.

***Corresponding author Email:** dengrui@hrbeu.edu.cn

If $|E| < U_V$, that means the combinations of all the errors in the D and S are less than U_V, and the validation is completed. If $U_V << |E|$, the calculation model needs to be improved.

Usually, a coefficient r is needed to refine the calculation grid in the verification, and it is defined as follows:

$$r_k = \frac{\Delta x_{k2}}{\Delta x_{k1}} = \frac{\Delta x_{k3}}{\Delta x_{k2}} = \ldots = \frac{\Delta x_{km}}{\Delta x_{k(m-1)}} \quad (6)$$

where r_k is the coefficient for the number k parameter, and Δx_k is the variation of it. As a rule, r is suggested to be $\sqrt{2}$. Denote the results with the finest grid as S_{k1}, S_{k2} and S_{k3}, the convergence coefficient R_k is:

$$R_k = \frac{\varepsilon_{k_{21}}}{\varepsilon_{k_{32}}} = \frac{S_{k2} - S_{k1}}{S_{k3} - S_{k2}} \quad (7)$$

The calculation results can be classified into three types based on R_k:

① $0 < R_k < 1$: the calculation result is of monotonic convergence;

② $R_k < 0$: the calculation result is of oscillatory convergence;

③ $R_k > 1$: the calculation result is of divergence.

3 Discussion of grid generation for catamaran resistance calculation

It is known that, in an idiographic problem, grid generation affects the result greatly. Although the research about uncertainty is important, research about the scheme and some parameters in grid generation does have its engineering significance. For the simulation of a ship flow, a fine grid is needed around the ship because velocity grads near the ship surface is great, and also it involves free surface, so that it is a big problem between grid quality and result precision. Usually, for an engineering problem, a balance is needed.

In this paper, the flow of a catamaran is calculated. The length on water line L_{wl}, width and depth are respectively 2.71, 0.15 and 0.127 metres, the area of wetted surface is 1.77 square meters, the distance between the demihulls is 0.65 metres. It is shown in Fig.1.

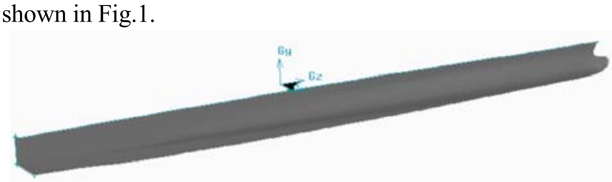
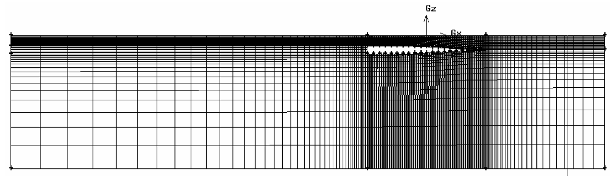
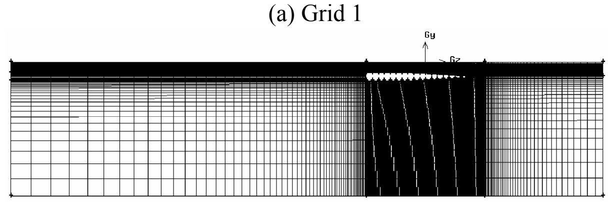


Fig.1 One demihull of the catamaran

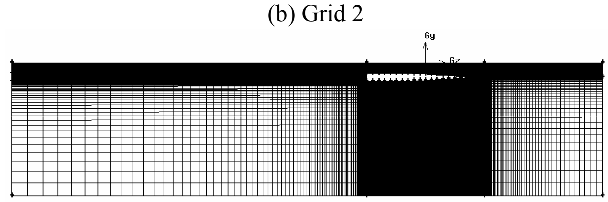
Using $r = \sqrt{2}$, generate Grid 1, Grid 2 and Grid 3 with the number of grids of $198 \times 28 \times 16$, $286 \times 40 \times 22$ and $396 \times 56 \times 32$, respectively. The grid is shown in Fig.2.



(a) Grid 1



(b) Grid 2



(c) Grid 3

Fig.2 Grid generation

The calculation speed is 4.6m/s and the resistance is calculated. The results are shown in Table1. Where S_1, S_2 and S_3 are the results with Grid 1, Grid 2, and Grid 3, $\varepsilon_{23} = S_2 - S_3$, $\varepsilon_{12} = S_1 - S_2$, $R = \varepsilon_{12} / \varepsilon_{23}$. The result $R < 1$ shows Grid 3 can be used for calculation, but the difference between calculation result of Grid 3 and test data is about 4.5% and there're only 396×84 nodes on the free surface, so the wave far from catamaran can not be calculated well by VOF method, and the method of grid generation needs to be improved.

Table1 Calculation result

S_1	S_2	S_3	ε_{23}	ε_{12}	R
42.43	41.59	40.63	0.96	0.84	0.875

The free surface of Grid 3 is shown in Fig.3.

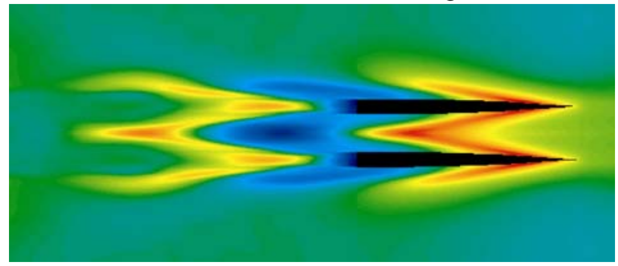


Fig.3 Free surface

In the research of grid generation, three elements must be taken into account: the size of the grid on the ship surface, the distance between the first grid node and ship surface and the coefficient used to refine the grid.

The grids used to mesh the ship surface have sizes of 8 mm, 12 mm, 16 mm and 20 mm respectively, i.e. 3.0‰, 4.4‰, 6.0‰ and 7.4‰ of L_{wl}, and the total grid numbers for the flow field are correspondingly 1.9, 0.54, 0.26 and 0.12 millions. When the 8 mm grid is used, the quantity of the grids reaches the maximum one which exceeds the number that a computer can hold; and when the 20 mm grid is used, there are fewer grids for the ship. The calculation speed is 4.6 m/s, the resistance changing with time steps is shown as follows, where l means the size of the grid.

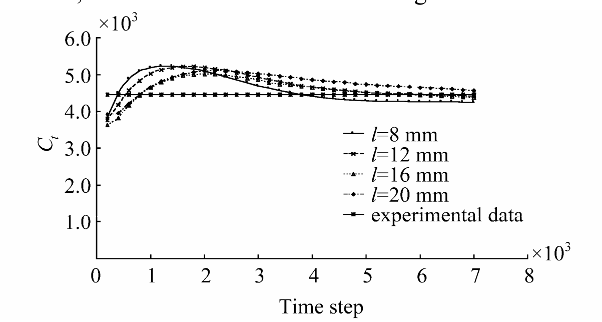


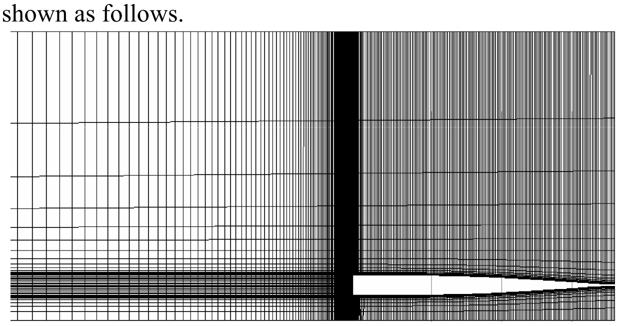
Fig.4 Ship resistance of different mesh size on ship surface

Comparing the results, one can find that the result with the grid sized 8mm converges faster than the others, but it crosses the test data line and the discrepancy increases with time steps. The result with the grid sized 20 mm shows a poor precision. The results of the rest two seem similar, but meshing the ship surface with the grid sized 16 mm generates fewer grids for the flow field, so the mesh size on the catamaran surface is suggested to be about 6.0‰ of L_{wl}.

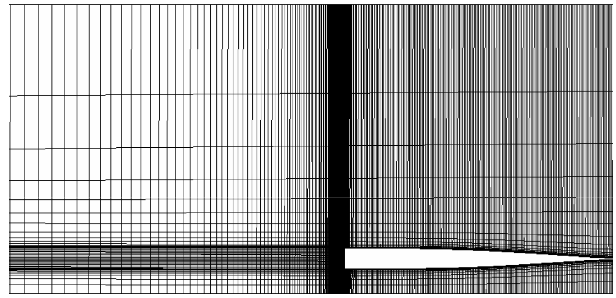
The first node distribution decides the parameter y^+, which is defined as follows (Cheng, Scott Percival and Eugene H. Gotimer, 1995):

$$y^+ = 0.172\left(\frac{y}{L}\right)Re^{0.9} \tag{8}$$

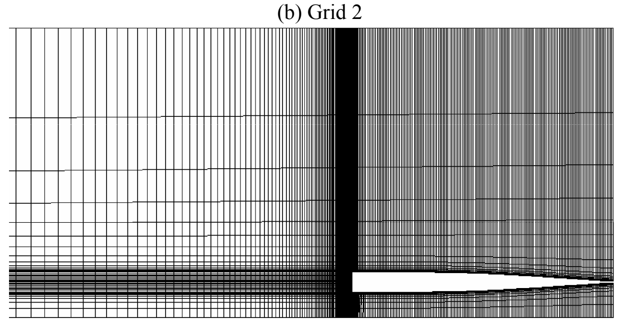
Where y is the distance between the first node and the ship surface, L is the length of the ship, Re is Reynold number. Another coefficient r^* is needed to distribute the nodes. Discretizing the flow field with the grid, taking $r^* = \sqrt{2}$ and y as 0.4 mm, 0.8 mm and 1.6 mm, respectively, generate Grid 1, Grid 2 and Grid 3. Keeping the same distance, but using a smaller r^*, about 1.06, to generate Grid 4, Grid 5 and Grid 6. The grid distribution is shown as follows.



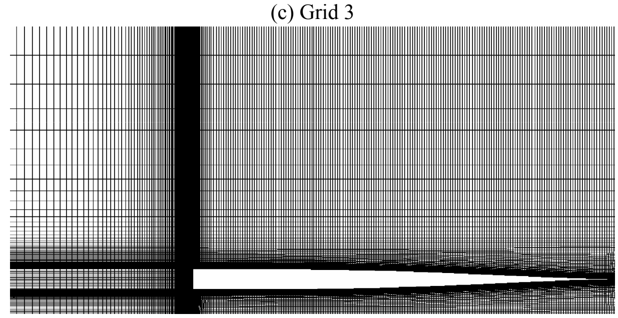
(a) Grid 1



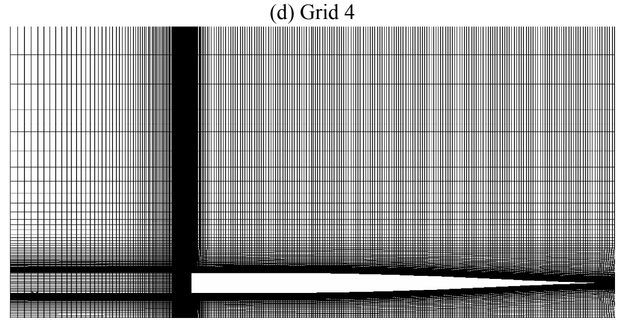
(b) Grid 2



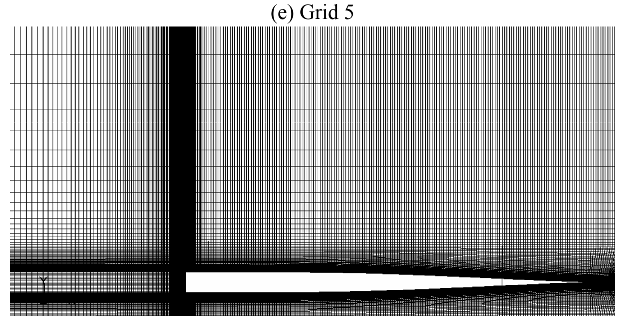
(c) Grid 3



(d) Grid 4



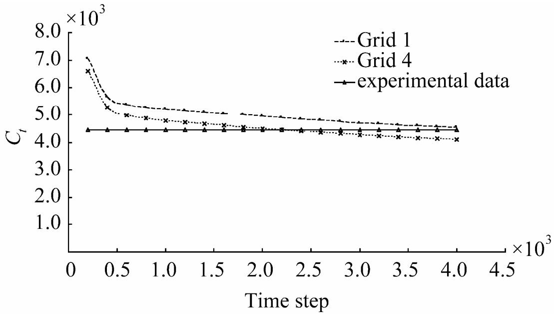
(e) Grid 5



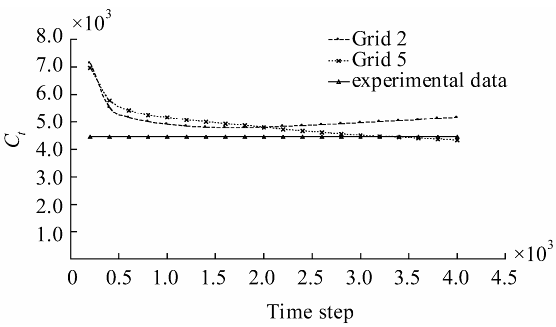
(f) Grid 6

Fig.5 Grid distribution of flow field with different first boundary layers

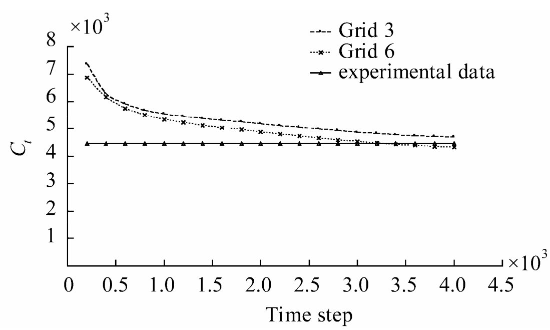
The calculated ship speed is 4.6 m/s, the resistance of the first 4000 time steps is shown as follows.



(a) y = 0.4mm



(b) y = 0.8mm



(c) y = 1.6mm

Fig.6 Comparison of the results with different grid distributions

Using different r^* gives different results, and the difference between them can not be ignored. In some conditions, using a large r^* may produce a divergent result. Comparing the results, it shows that the grid generated by taking $r^*=\sqrt{2}$ is a litter large, the result could not get a correct precision; an r^* less than $\sqrt{2}$ may fit the flow field better.

Taking $r^*\approx1.06$ for example, it is quite less than $\sqrt{2}$, and at this time y = 0.4 mm, 0.8 mm, 1.0 mm, 1.2 mm, 1.6 mm and 2.0 mm respectively, at that time, y^+ is between 56.2 and 281.2, the flow field is meshed. When y is chosen to be 0.4 mm, the grid number appears too great, so a smaller y should be avoided. For the same speed 4.6 m/s, the resistance values within the first 4000 time steps are shown.

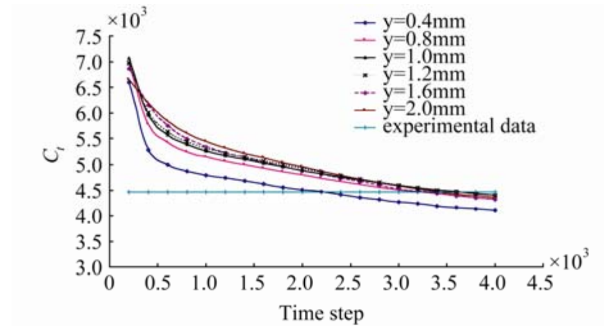


Fig.7 Ship resistance values with different distance of the first layer of the nodes off the hull surface

It is shown in the comparison that the difference between the results associated with $y=0.8\sim2.0$ mm can be ignored, but the numerical result with y =0.4 mm is smaller than the others, and it is far below the experimental data. The reason is likely that the node is already in the viscous sublayer, which can not lead to a correct result. Considering the other research, for a catamaran, the suggested range of y^+ is better to be within $100\leqslant y^+\leqslant200$. A smaller y^+ may put the node in the viscous sublayer, and a larger one may generate a coarse grid, which may lead to poor calculation result of the flow field.

4 Resistance calculation of catamarans

Resistance of a catamaran is calculated. The calculated ship's Fr numbers are 0.39, 0.48, 0.58, 0.68, 0.78, 0.83, 0.89, 0.95, 1.03, and 1.12. Comparison of the numerical and the test data is shown in Fig.8.

It shows a good agreement. The free surface is shown in Fig.9.

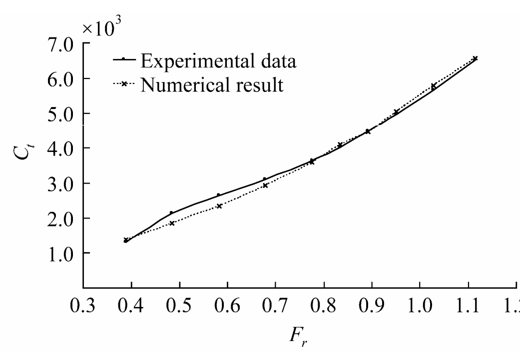


Fig.8 Ship resistance comparison between the calculation and test data

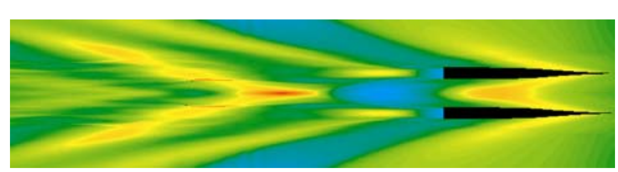


Fig.9 Free surface with new grid distribution

5 Conclusions

In the discussion of resistance calculation of the catamaran, some conclusions are gotten:

1) A grid size about 6.0‰ of L_{wl} can be used as the size for meshing the catamaran surface;

2) A smaller coefficient r^*, smaller than $\sqrt{2}$, is suggested to refine the grid of the flow field;

3) The nondimensional distance of the first layer of the nodes from the hull surface, y^+, is quite important. For catamaran resistance calculation, the range of y^+ should better to be $100 \le y^+ \le 200$, but a smaller r^* is suggested at the same time.

The research in this paper only includes a few factors in the numerical simulation, though that may be useful for the resistance calculation, and further research is thirsted for improving it.

Reference

Chen Jingpu,; Zhu Dexiang; He Shu-long (2006). Research on numerical prediction method for wavemaking resistance of catamaran/trimaran. *Journal of Ship Mechanics*, **10**(2), 23-29.

Chen Kang, Huang Debo; Li Yunbo (2008). Grid convergence study in the resistance calculation of a trimaran. *Journal of Marine Science and Application*, **7**, 174-178

Cheng Wen-Lin, Scott P, Eugene HG (1995). Viscous drag calculations for ship hull geometry. http://conan.dt.navy.mil/reports/viscous/viscous.pdf.

Deng Rui, Huang Debo (2007). Numerical simulation of hydrodynamic forces on a slant strut Small Waterplane Area Twin Hull with bow fins. *Journal of Ship Mechanics*, **11**(4), 564-567.

Fred S, Robert VW, Hugh WC, (1999). Verification and validation of CFD simulations. IIHR(Iowa Institute of Hydraulic Research) Report, No 407. University of Iowa.

ITTC QM Procedure (2002). 7.5- 03- 01- 01.

ITTC QM Procedure (2002). 7.5- 03- 02- 01.

Larsson L, Stern F, Bertram V (2003). Benchmarking of computational fluid dynamics for ship flows: the Gothenburg 2000 Workshop. *J Ship Research*, **47**(1), 63-81.

Roache PJ (1998). *Verification and validation in computational science and engineering*. Albuquerque, Hermosa publishers, New Mexico.

Simonsen CD, Stern F (2003). Verification and validation of RANS maneuvering simulation of Esso Osaka: effects of drift and rudder angle forces and moments. *Computers and Fluids*, **32**, 1325-1356.

Stern F, Wilson RV, Coleman HW, Paterson EG (2001). Comprehensive approach to verification and validation of CFD simulations-Part I: methodology and procedures. *J Fluids Engineering*, **123**, 793- 802.

Tao Wen-quan (2001). *Numerical Heat Transfer(Second Edition)*. Xi'An JiaoTong University Press, Xi'An.

Wilson R, Stern, F (2002). Verification and validation for RANS simulation of a naval surface combatant. Standards for CFD in the Aerospace Industry. *Aerospace Sciences Meeting*, Reno, Nevada, AIAA 2002-0904.

Zhao Fuyun; Tang Guangfa (2004). Feedback of systematic errors and implementation in CFD numerical simulation. *Hv & Ac*, **34**(6), 1-8.

Zhu Dexiang; Zhang Zhirong; Wu Chengsheng; Zhao Feng (2007). Uncertainty analysis in ship CFD and the primary application of ITTC procedures. *Journal of Hydrodynamics(Ser.A)*, **22**(3), 363-370.

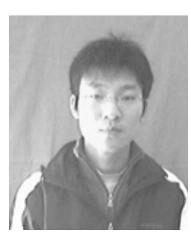
Zou Zaojian; Luo Qingshan; Shi Yiming (2005). A research on resistance prediction of SWATH ships. *Shipbuilding of China*, **168**(1), 14- 21.

Rui Deng was born in 1981. He is a PhD candidate at Harbin Engineering University. His current research interests include ship hydrodynamics and resistance.

De-bo Huang was born in 1943. He is a professor at Harbin Engineering University. His current research interests include ship hydrodynamics and resistance, high performance vessel, optimization etc.

Jia Li was born in 1982. She is a PhD candidate at Harbin Engineering University. Her current research interests include ship hydrodynamics and resistance.

Xuan-kai Cheng was born in 1985. He is a master at Harbin Engineering University. His current research interests include ship hydrodynamics and resistance.



Lei Yu was born in 1986. He is a master at Harbin Engineering University. His current research interests include ship hydrodynamics and resistance.