DOI: 10.3969/j.issn.1673-3185.2017.02.003 **http:** // english.ship-research.com

Translated from: QIAN H, SONG K W, GUO C Y, et al. Influence of waterjet duct on ship's resistance performance[J]. Chinese Journal of Ship Research, 2017, 12(2): 22-29.

Influence of waterjet duct on ship's resistance performance

QIAN Hao[1], *SONG Kewei*[2], *GUO Chunyu*[2], *GONG Jie*[2]

1 Marine Design and Research Institute of China, Shanghai 200011, China

2 School of Shipbuilding Engineering, Harbin Engineering University, Harbin 150001, China

Abstract: The waterjet duct can change the flow field of the stern, and it has a great influence on the resistance performance of the ship. The resistance performance of marine vehicles driven by waterjets is very different from that of conventional ships, so it is meaningful to study the changes to the resistance performance of the ship. We used the CFD software STAR-CCM+, treated the waterjet duct as the appendage and compared the change of the flow field in the stern after the installation of the waterjet duct at different angles. We described the change mechanism of the ship's resistance and resistance components by comparing the change in pressure distribution of the waterjet duct's surface and the flow field around the hull. The results show that STAR-CCM+ can realize the prediction of ship resistance performance because the simulation results achieved perfect accuracy, and it is gradually becoming the development direction of the resistance performance prediction of marine vehicles driven by waterjets. The installation of the waterjet duct will increase the resistance of the ship, which is mainly due to the increase of pressure resistance. In addition, the resistance performance of a ship driven by waterjets can be improved by the optimization of the waterjet duct's angle.

Key words: waterjets; ship resistance; numerical simulation; duct

CLC number: U661.31+1

0 Introduction

As a special propulsion mode, waterjet propulsion utilizes the reacting force of high-velocity flow ejected from the pump to push forward the ships[1]. Compared with traditional propeller propulsion, waterjet propulsion has more advantages, such as high efficiency of propulsion, low underwater noise, good adaptability to working condition[2] and good cavitation performance under high velocity[3].

The foreign research on waterjet began earlier. In recent years, CFD technology has gradually become a powerful research tool. On one hand, it obtains a favorable effect on studying the internal flow of complex pump. For example, the institutions such as National Maritime Technology Research Institute in Japan, Taylor Pool and University of Lowa in America all used CFD technology to conduct viscous flow field simulation analysis on the whole system of "ship + pump"[4]. Terwisga[5] from Delft University of Technology also studied the interaction between waterjet and hull. On the other hand, the current research focus is to use CFD software to realize the prediction of ship resistance performance. Brizzolara et al.[6] realized the prediction of the resistance performance of different types of trimaran by using CFD software and studied the structural layout of trimaran. Carr[7] analyzed the mutual wave-making interference and the generating cause between trimaran's main body and demihull, and reasonably optimized their relative positions via CFD numerical simulation. Mizine et al.[8] compared the change of flow field

Received: 2016 - 07 - 04

Supported by: National Natural Science Foundation of China (51209048, 41176074, 51409063); High-tech Ship Foundation of Ministry of Industry and Information Technology (G014613002); Young Backbone Teacher Support Program of Harbin Engineering University (HEUCFQ1408)

Author(s): QIAN Hao, male, born in 1980, senior engineer. Research interests: overall research and design of ships. E-mail: qianhao1125@126.com

SONG Kewei (Corresponding author), male, born in 1991, master candidate. Research interest: waterjet propulsion. E-mail: 1150811986@qq.com

of trimaran's stern through the combination of a lot of experimental data and CFD computation results, and further realized the optimization of the layout of trimaran's demihull.

The domestic research on waterjet started in the 1970s. Gao[9], Ge[10] and Yu[11] respectively studied the waterjet propulsion technology from different aspects. Yu[11] studied the effect of stern shape, duct parameter and waterjet on trimaran performance through CFD numerical simulation. Ding et al.[12] optimized the design of waterjet inlet duct by utilizing the computational fluid dynamics. Liu et al.[13] studied the effect of the size of flow control volume on the prediction of waterjet performance. Mao et al.[14] studied the effect of boundary layer on the flow field in the inlet duct of waterjet by utilizing CFD software FLUENT, which provided reference basis for the design of inlet duct.

The main part of waterjet that generates frictional resistance is the inlet duct which exerts great influence on hull resistance performance. The domestic studies on the inlet duct of waterjet are not many and the research methods are single. Therefore, this paper studies the influence of inlet duct on ship resistance performance from another angle. It takes waterjet duct as an appendage, compares the change of ship resistance before and after the installation of waterjet duct and elaborates the changing mechanism of ship resistance and resistance component by comparing the pressure distribution of stern and the change of hull streamline, so as to provide new idea and reference basis for studying the influence of waterjet duct on ship resistance performance.

1 CFD numerical simulation

1.1 Establishment of calculation model

This paper uses CATIA software for modeling, optimizes the layout of FA1-type trimaran and then gets the calculation model. The basic parameters of the calculation model are shown in Table 1.

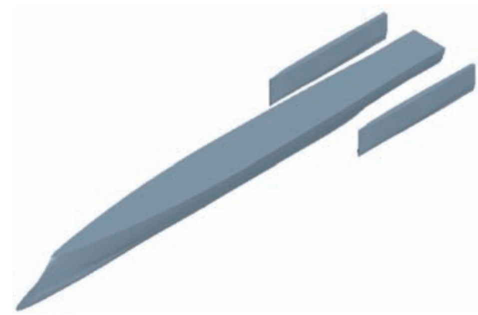
Table 1 Basic parameters of the calculation model

Main parameters	Numerical values
Total length of main body/m	3.501
Total length of demibody/m	0.988
Moulded depth/m	0.279
Draft/m	0.182
Total breadth moulded /m	0.791

This paper selects three inlet ducts with different dip angles (25°, 30° and 40°) so as to avoid the error caused by the test oneness and optimize the waterjet duct.

The calculation model and waterjet duct model are respectively shown in Fig. 1 and Fig. 2.

(a) Front view



(b) Side view

Fig.1 CATIA modeling

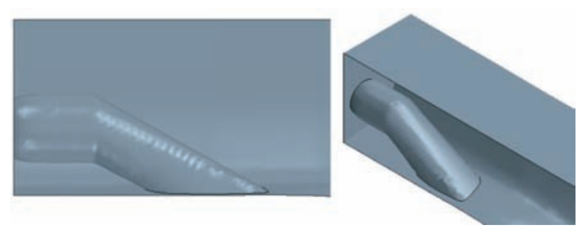


Fig.2 Waterjet duct modeling

As for the selection of computational domain, this paper selects the computational domain of half a ship. By consulting relevant literature[15], the computational domain is concretely set as:

1) One time of ship length is selected in front of the ship;

2) Two times of ship length are selected in the rear of the ship;

3) 0.5 time of ship length is selected above the waterline plane;

4) One time of ship length is selected below the waterline plane;

5) 1.5 times of ship length are selected in the direction of ship breadth.

The computational domain model is shown in Fig. 3.

1.2 Mesh division

Mesh division is an important part of the whole numerical simulation process which has a lot of work to do and is the most time-consuming. The quantity

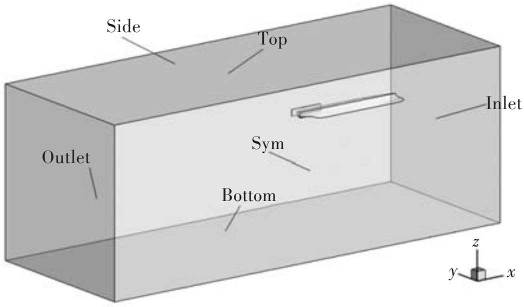
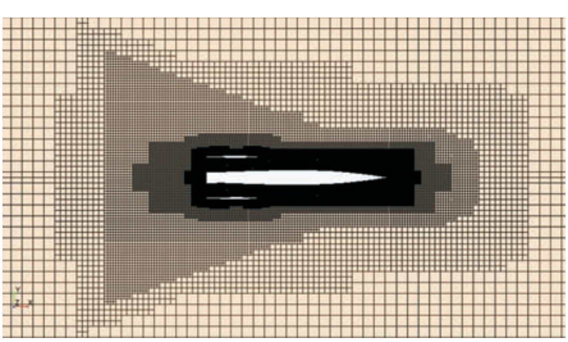


Fig.3 Computational domain

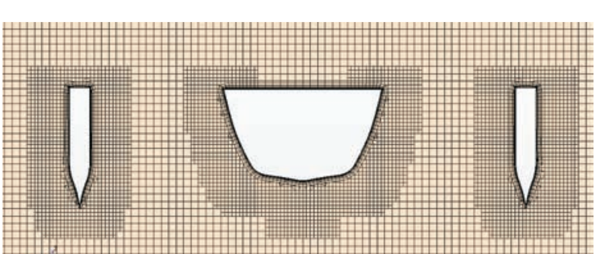
and quality of meshes will exert an important influence on computation result. So, the meshes cannot be divided too dense or too sparse. If the meshes are too dense, the computing time will be increased and the accuracy of computation will not be significantly improved. Sometimes, an exactly opposite effect may be obtained. If the meshes are too sparse, the accuracy degree of computation result usually can't meet the requirement.

The curvature in the bow and stern of ship changes greatly. So it must be separately densified so as to guarantee the mesh quality. Meanwhile, Kelvin wave system is very important for the study of wave-making resistance of ship. Therefore, it is needed to conduct mesh densification on the region near the hull (Fig. 4). In this paper, the densified meshes are generally divided into three transition layers so as to guarantee the mesh quality.

Before mesh division, the thickness of boundary layer δ should be firstly calculated, which is directly related to the division of the meshes near the wall and exerts a great influence on the accuracy of final computation result.



(a) Mesh division of Kelvin wave system



(b) Mesh division of transverse section

Fig.4 Mesh division graph

$$Re_\delta = 0.14 Re_x^{6/7} \tag{1}$$

$$\delta = 0.14 L Re_x^{6/7} \frac{1}{Re_L} \tag{2}$$

where, x is the distance to the bow; L is the ship length; δ is the thickness of boundary layer; Re is Reynolds number; Re_x, Re_L and Re_δ are the corresponding Reynolds numbers.

For "the blunt body" like ship, the boundary layer of Re_x is not calculated from zero at the stagnation point. So, for the sake of security, Re_δ is generally set as 20%–25% of Re_L, which is set as 20% in this paper. The empirical formula of boundary layer thickness is:

$$\delta = 0.028 L Re_x^{-1/7} \tag{3}$$

Mao et al.[14] believed that the boundary layer of inlet exerts a certain influence on inlet duct. By referring to the obtained conclusion, the meshes of boundary layer are divided, with seven boundary layers.

1.3 Selection of physical model

Nowadays, the simulation on free surface includes Marker and Cell (MAC) and Volume of Fluid (VOF). Compared with VOF method, MAC method has many disadvantages. For example, it has a large amount of calculation and is easy to distort in some cases. VOF method is the most commonly used method at present, which mainly originates from Hirt and Nichols's idea and brings international research fever.

VOF method is used to study the interface of two or multiple incompatible media. The sum of volume fractions of all media is 1. VOF method is realized through network volume fraction function f whose value expresses the fluid proportion. This paper studies two fluid media which are air and water. The appointed fluid phase is air.

Besides the selected VOF method, the turbulence model is selected as $k-\varepsilon$ model. The gravity model, element mass correction and VOF wave are selected from the optional models.

1.4 Setting of boundary condition

The boundary condition is generally divided into two kinds: the first kind is permeable boundary such as velocity inlet. The material interchange of incoming flow can occur at these boundaries. The second kind is non-permeable boundary like solid surface. Accordingly, the material interchange will not occur at these boundaries. During CFD numerical calculation, the boundary condition is generally divided into the following kinds: inlet boundary, outlet boundary, boundary of symmetric plane and boundary of solid surface.

When setting the boundary condition, this paper just sets the hull and its appendage as the wall surface boundary and does not set the computational domain boundary as wall surface, which can more truly simulate the broad water area of the ship in the actual voyage and get more accurate computation result.

Finally, the boundary condition type of computational domain is set as follows: the inlet type is velocity inlet; the outlet type is pressure outlet; the midship section is symmetric plane; the hull and duct are set as non-slipping wall surface.

The specific setting of boundary condition of computational domain is shown in Fig. 3.

2 Resistance performance prediction of bare hull

2.1 Comparative analysis of resistance value and experimental value

As shown in Table 2 and Fig. 5, when the navigational speed of the ship is low, the difference between calculated value of resistance and experimental value for bare hull is small. The experimental value and calculated value coincide well. When *Fr* is 0.103, their difference is 3.446%. When *Fr* is between 0.103 and 0.441, their difference maintains at about 5%, which indicates a good computational accuracy.

However, with the increase of navigational speed, their difference value also increases. When *Fr* is 0.485, the difference value reaches 9.207%, which is mainly caused by the slight difference between the process of simulated calculation and physical environment in the test. The ship model may pitch or

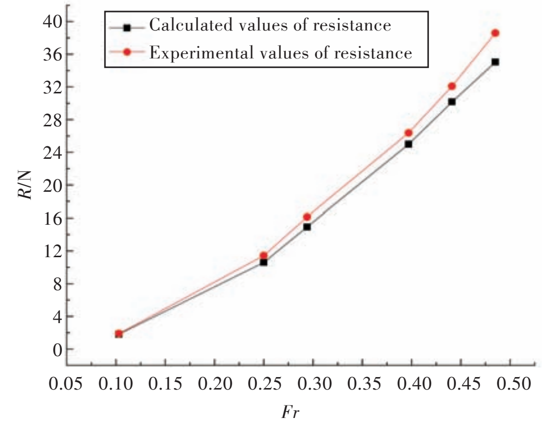


Fig.5 Comparison of resistance and experimental values for bare hull

heave in high-speed test, which will exert a great influence on ship resistance. With the increase of navigational speed, the increase rate of resistance value gets larger. The navigational speed greatly affects the resistance value.

2.2 Comparison of resistance component under different navigational speeds

This paper divides the hull resistance into shear resistance and pressure resistance for analysis. The shear resistance is the resultant force of shear stress, namely, frictional resistance. The pressure resistance is the sum of viscous pressure resistance and wave-making resistance.

It can be seen from Table 3 and Fig. 6 that when the navigational speed of the ship is low, the ratio of shear resistance (frictional resistance) to the total resistance of ship is large. When *Fr* is 0.103, the shear resistance approximately accounts for two thirds of total resistance. With the increase of navigational speed, the shear resistance and pressure resistance gradually tend to be equal. This is mainly because the ship wave-making resistance and its proportion in total resistance gradually increase, with the increase of navigational speed.

Table 2 Comparison of resistance and experimental values

Froude number	Calculated values of resistance/N	Experimental values of resistance/N	Error values/%
0.103	1.821	1.886	−3.446
0.250	10.597	11.439	−7.361
0.294	14.934	16.150	−7.529
0.397	24.981	26.367	−5.257
0.441	30.171	32.078	−5.945
0.485	35.045	38.599	−9.207

Table 3 Comparison of each resistance component

Froude number	Shear resistance/N	Pressure resistance/N	Proportion of shear resistance/%
0.103	1.210	0.610	66.4
0.250	5.773	4.823	54.5
0.294	7.721	7.213	51.7
0.397	13.461	11.519	53.9
0.441	16.136	14.034	53.5
0.485	18.855	16.190	53.8

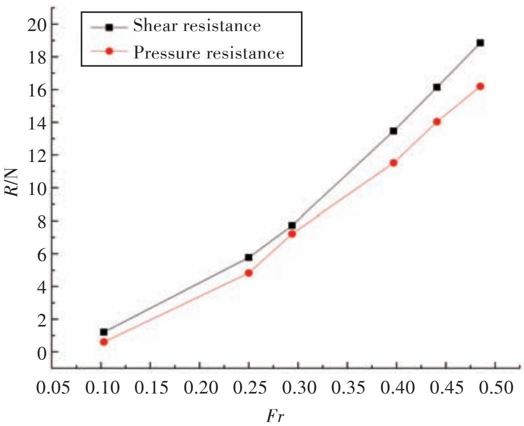


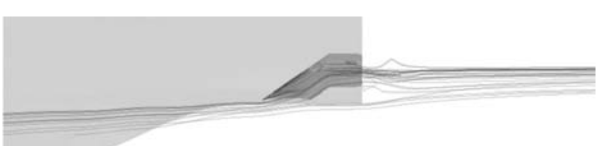
Fig.6 Resistance components

3 Influence of waterjet duct on ship resistance performance

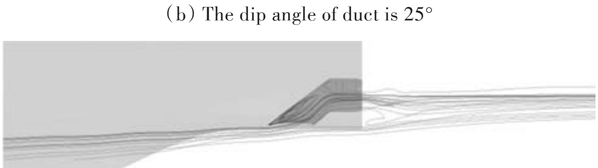
3.1 Comparative analysis of streamline result

Fig. 7 and Fig. 8 respectively present the comparison of streamline results when Fr equals 0.103 and 0.485. It can be seen from the figures that the installation of waterjet duct will change the movement trajectory of flow. The flow will change the original flow direction in the inlet duct and even form vortex and backflow which will reduce the flow's kinetic energy and increase ship resistance. By comparing the streamline diagrams of the hull installed with duct at

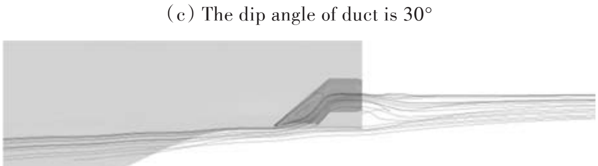
(a) Bare hull



(b) The dip angle of duct is 25°



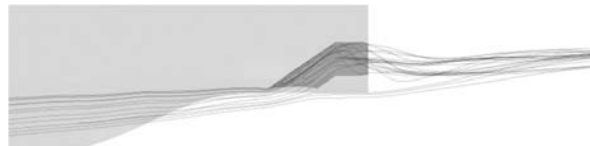
(c) The dip angle of duct is 30°



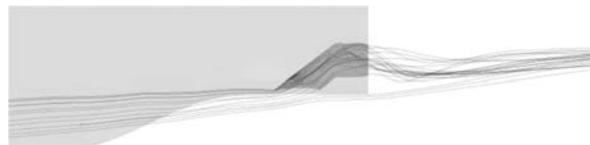
(d) The dip angle of duct is 40°

Fig.7 Streamline diagram at Fr=0.103

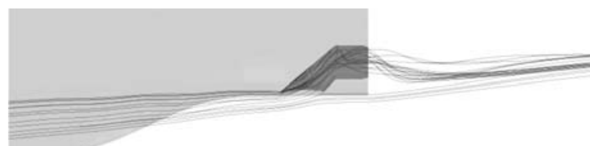
(a) Bare hull



(b) The dip angle of duct is 25°



(c) The dip angle of duct is 30°



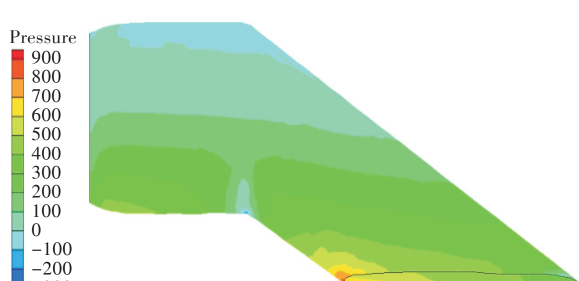
(d) The dip angle of duct is 40°

Fig.8 Streamline diagram at Fr=0.485

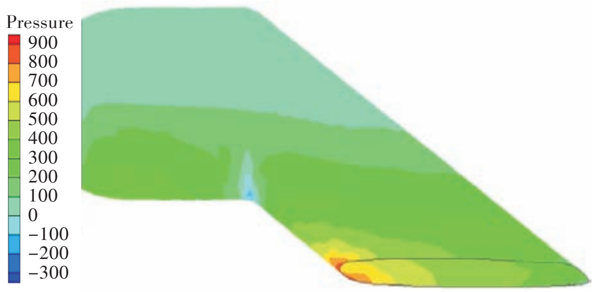
different inflow angles, it can be found that with the increase of duct dip angle, the fluid direction in the inlet duct changes greatly. It can be seen from the figure that when the duct dip angle is 40°, the backflow will occur inside the waterjet, which is adverse to ship resistance performance.

3.2 Comparative analysis of duct surface pressure

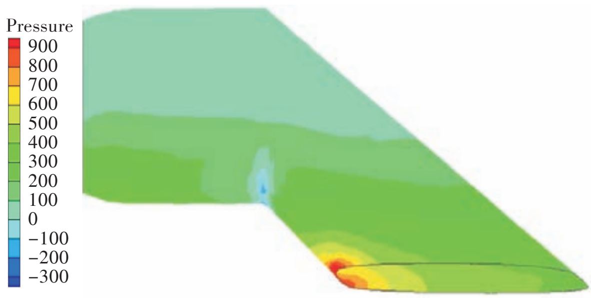
Fig. 9 and Fig. 10 are respectively the comparison diagrams of duct surface pressure when Fr equals 0.294 and 0.485. It can be seen from the figures that regardless of whether Fr is 0.294 or 0.485, the surface pressure distribution of waterjet duct is uneven. At the junction of water inlet and lower wall surface, the pressure is the largest and the speed is the lowest. While, at the inflection point of lower wall surface, the pressure is the lowest and the speed is the largest. So vortex or backflow easily occurs at this place.



(a) The dip angle of duct is 25°

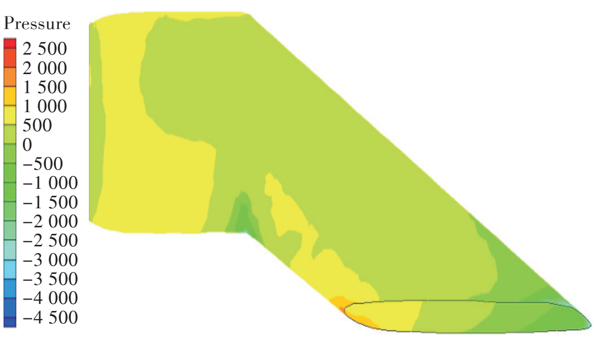


(b) The dip angle of duct is 30°

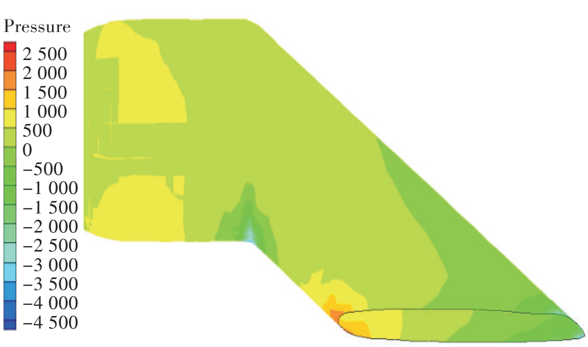


(c) The dip angle of duct is 40°

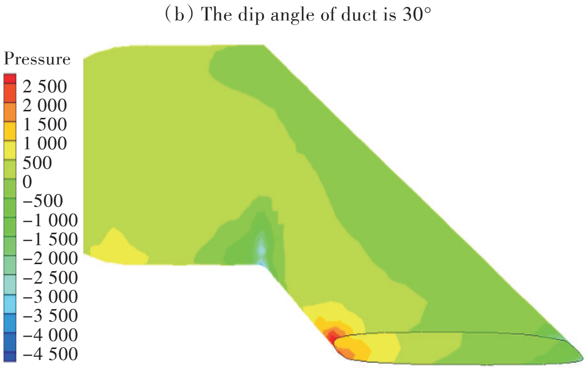
Fig.9 Pressure distribution of duct surface at Fr=0.294



(a) The dip angle of duct is 25°



(b) The dip angle of duct is 30°



(c) The dip angle of duct is 40°

Fig.10 Pressure distribution of duct surface at Fr=0.485

By comparing the surface pressure distribution of waterjet under different navigational speeds, it can be found that the pressure difference will increase with the increase of navigational speed. By comparing the surface pressure distribution of waterjet under the same navigational speed, it can be found that the pressure value at the inflection point of lower wall surface gradually decreases while the pressure at the junction of water inlet and lower wall surface increases with the increase of duct dip angle. Therefore, it is more likely to produce vortex or backflow, which is adverse to ship resistance performance.

3.3 Comparative analysis of hull resistance value

Table 4 shows the total resistance and resistance component values of the ship under different working conditions. Fig. 11 shows the resistance and the increase of resistance.

It can be seen from the figure that:

1) After the installation of waterjet duct, the total resistance of the ship increases. When the navigational speed of the ship is low, the resistance value of the ship driven by waterjets increases slightly. When Fr is between 0.103 and 0.294, the resistance increment is between 4% and 5%. When the ship travels at high speed and Fr is 0.485, compared with the bare hull, the resistance value of the ship driven by waterjets increases by 10% to 12%.

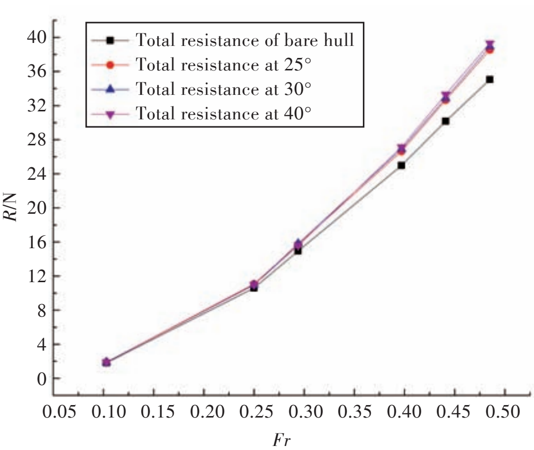
2) By comparing the total resistance value of ship installed with waterjet ducts with different duct dig angles, it can be found that the difference of resistance value is not large at low speed. But with the increase of navigational speed, the resistance value is relatively small when the duct dip angle is 25°, namely, the waterjet duct exerts the least influence on ship resistance. After the installation of waterjet duct, the hull resistance values change and have the same tendency.

3) After the installation of waterjet duct, the hull shear resistance changes slightly. With the increase of navigational speed, the increment percentage of ship shear resistance changes slightly and maintains at about 0.5% to 2%. That's mainly because the area of inlet duct and hull surface area have a large difference. After the installation of waterjet duct, the wetted surface area of the ship increases by about 0.9%. The shear resistance changes less than the pressure resistance.

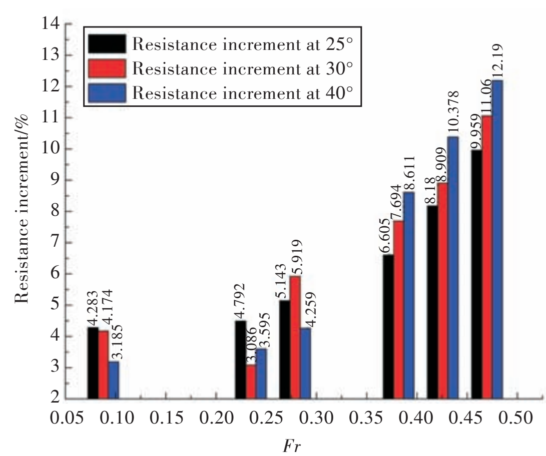
4) The installation of waterjet duct greatly affects the ship pressure resistance. When the navigational

Table 4 Resistance values under different working conditions

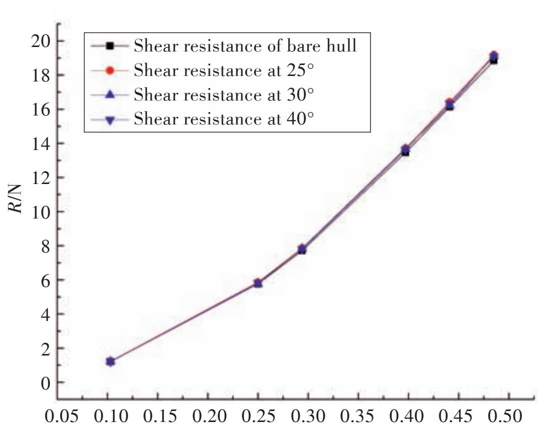
Froude number	Hull	Total resistance/N	Shear resistance/N	Pressure resistance/N
0.103	Bare hull	1.821	1.210	0.610
	25°	1.899	1.222	0.678
	30°	1.897	1.220	0.677
	40°	1.889	1.214	0.675
0.250	Bare hull	10.597	5.773	4.823
	25°	11.073	5.853	5.220
	30°	10.954	5.776	5.178
	40°	10.978	5.831	5.147
0.294	Bare hull	14.934	7.721	7.213
	25°	15.072	7.854	7.848
	30°	15.818	7.833	7.985
	40°	15.557	7.804	7.753
0.397	Bare hull	24.981	13.461	11.519
	25°	26.631	13.696	12.935
	30°	26.903	13.672	13.231
	40°	27.132	13.629	13.503
0.441	Bare hull	30.171	16.136	14.034
	25°	32.639	16.421	16.218
	30°	32.859	16.210	16.649
	40°	33.302	16.346	16.957
0.485	Bare hull	35.045	18.855	16.191
	25°	38.535	19.167	19.368
	30°	38.921	19.126	19.794
	40°	39.317	19.081	20.236



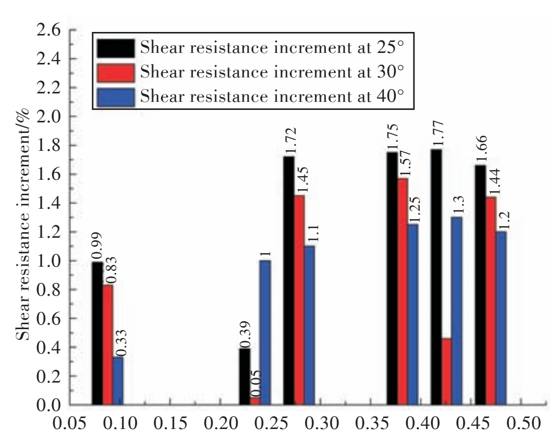
(a) Comparison of total resistance



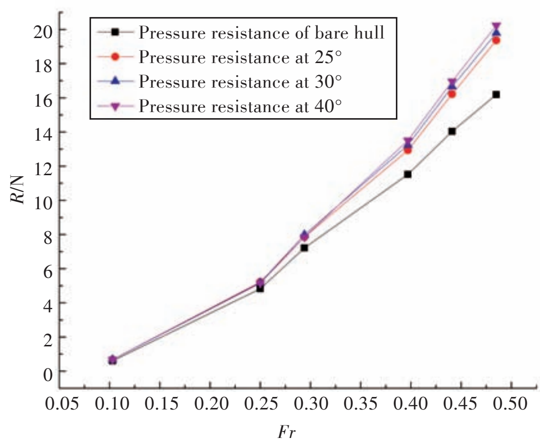
(b) Total resistance increment



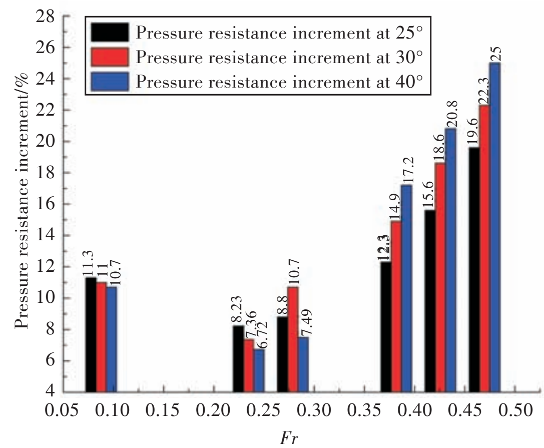
(c) Comparison of shear resistance



(d) Shear resistance increment



(e) Comparison of pressure resistance



(f) Pressure resistance increment

Fig.11 The changes of resistance and the resistance components

speed is low, the pressure resistance increment is relatively small and maintains at about 10%. With the increase of navigational speed, the pressure resistance increment of ship is larger than that of bare hull after the installation of waterjet duct. When *Fr* is 0.485, the hull resistance of waterjet duct with dip angle of 40° increases by 25%. That's because the installation of waterjet duct will affect the flow field in the stern, increase the pressure difference between bow and stern and further increase the hull pressure resistance.

By comparing the pressure resistance under three different situations, it can be found that the pressure resistance value is the smallest when the duct dip angle is 25°, which is the same as the change situation of total resistance.

4 Conclusions

This paper uses STAR-CCM+ software to conduct numerical simulation on the improved FA1-type trimaran, compares the change of flow field of the stern after waterjet ducts with different inflow angles are installed and elaborates the changing mechanism of resistance and resistance component by comparing the change of hull streamline and duct pressure distribution. The conclusions are as follows:

1) STAR-CCM+ can realize the prediction of ship resistance performance and gradually becomes the development direction of the prediction of resistance performance of a ship driven by waterjets. Through the comparision between calculated value and experimental value of the bare hull resistance, it can be found that when *Fr* is 0.103 to 0.441, their difference value maintains at about 5%, which reaches a good computational accuracy.

2) The installation of waterjet duct can increase ship resistance, mainly resulting from the increase of pressure resistance. By comparing the resistance values of bare hull and the hull installed with waterjet duct, it can be found that when *Fr* is 0.485, the total resistance of the hull increases by 11.1%, and the shear resistance and pressure resistance respectively increase by 1.5% and 22%, which illustrates that the change of resistance is mainly caused by the increase of pressure resistance.

3) The optimization of duct dip angle can improve the resistance performance of a ship driven by waterjets. Within certain range, the increase of duct dip angle will increase the ship resistance. When *Fr* is 0.485 and the duct dip angle is 40°, the total resistance value is 2.03% higher than that when the duct dip angle is 25°.

This paper studies the influence of waterjet duct on ship resistance performance from the viewpoint of appendage. This research viewpoint and the obtained conclusions have a certain reference value. Another research viewpoint is to study the duct resistance through simulating the real working condition of waterjet, which is also the key content of the following work. It is expected to get more valuable research results by comparing the difference of two research viewpoints.

References

[1] LI X. Numerical simulation of flow field characteristics of the high-speed craft waterjet propulsion [D]. Zhenjiang: Jiangsu University of Science and Technology, 2013(in Chinese).

[2] LIU Z, MENG F L. The development on technology of water jet propulsion for ship[J]. Marine Technology, 2004(4): 42-44 (in Chinese).

[3] WANG L X. Ship & Boat, 1997(3): 45-52(in Chinese).

[4] SUN C L, WANG Y S. Computational fluid dynamics for marine waterjet design and performance analysis [J]. Journal of Harbin Engineering University, 2008, 29(5): 444-448 (in Chinese).
[5] VAN TERWISGA T. Waterjet-hull interaction [D]. Delft: Delft University of Technology, 1996.
[6] BRIZZOLARA S, BRUZZONE D. Hydrodynamic optimisation of high-speed trimaran hull forms [C]//Proceedings of the 18th International Offshore and Polar Engineering Conference. Vancouver, BC, Canada: ISOPE, 2008: 547-554.
[7] CARR B. Investigation of trimaran interference effects [D]. Glen Cove: Webb Institute, 2007.
[8] MIZINE I, KARAFIATH G, QUEUTEY P, et al. Interference phenomenon in design of trimaran ship [C]// Proceedings of the 10th International Conference on Fast Sea Transportation-FAST. Athens, Greece: Power and Rotary Systems, 2009.
[9] GAO S. Study on course/speed control of ship with waterjet propulsion [D]. Harbin: Harbin Engineering University, 2008 (in Chinese).
[10] GE Y L. Research on the relationship between stern shape and resistance of trimaran adopting water jet [D]. Harbin: Harbin Engineering University, 2009 (in Chinese).
[11] YU D W. The prediction of the flow domain and the resistance of trimaran adopting water-jet [D]. Harbin: Harbin Engineering University, 2011 (in Chinese).
[12] DING J M, WANG Y S. Parametric design and application of inlet duct of marine waterjet [J]. Journal of Shanghai Jiaotong University, 2010, 44 (10): 1423-1428 (in Chinese).
[13] LIU C J, WANG Y S, ZHANG Z H. Study on flow control volume in numerical simulation of waterjet propulsor [J]. Chinese Journal of Hydrodynamics, 2008, 23(5): 592-595 (in Chinese).
[14] MAO X F, TANG S L. Numerical study of boundary layer effects on waterjet intakes flow and pressure distribution [J]. Journal of Hydrodynamics, 2005, 20 (4): 479-485 (in Chinese).
[15] ZHANG Z R, ZHAO F, LI B Q. Application of k-ω turbulence model to computation of viscous flow field around a ship [J]. Journal of Ship Mechanics, 2003, 7(1): 33-37 (in Chinese).

喷水推进器流道对船舶阻力性能的影响

钱浩[1], 宋科委[2], 郭春雨[2], 龚杰[2]
1 中国船舶及海洋工程设计研究院,上海 200011
2 哈尔滨工程大学 船舶工程学院,黑龙江 哈尔滨 150001

摘　要:[目的] 喷水推进船舶的阻力性能与常规船舶有着很大的不同,喷水推进器流道的存在会改变船舶尾部流场,对船舶阻力性能有着很大的影响。[方法] 以FA1型三体船为计算模型,利用CFD软件STAR-CCM+,将喷水推进器流道看作附体,对比研究安装不同进流角喷水推进器流道前后船舶尾部流场变化。通过对比流道表面压力分布、船体流线的变化,阐述船舶阻力以及阻力成分产生变化的机理。[结果] 结果表明:STAR-CCM+可以实现对于船舶阻力性能的预报;喷水推进器进水流道的安装会增大船舶阻力,主要为压差阻力的增大。[结论] 对进水流道倾角的优化可以增进喷水推进船舶的阻力性能。

关键词: 喷水推进器; 船舶阻力; 数值模拟; 流道