Research Article

Investigation of Complex Characteristics of Waterjet Propulsion Device with Intake Grid

Can Luo ^{1,2}, Shuaihao Lei,¹ Chuan Wang,^{1,3} Li Cheng ¹, Chenzhi Xia,⁴ Di Zhang,¹ and Qiang Guo⁵

¹College of Hydraulic Science and Engineering, Yangzhou University, Yangzhou, Jiangsu 225009, China

²Key Laboratory of Fluid and Power Machinery of Ministry of Education, Xihua University, Chengdu, Sichuan 610039, China

³Hainan Vocational University of Science and Technology, Haikou, Hainan 571126, China

⁴The Eastern Route of South-to-North Water Diversion Project Jiangsu Water Source Co, Ltd, Nanjing, Jiangsu 210019, China

⁵College of Electrical, Energy and Power Engineering, Yangzhou University, Yangzhou, Jiangsu 225127, China

Correspondence should be addressed to Can Luo; luocan@yzu.edu.cn and Li Cheng; chengli@yzu.edu.cn

Received 20 November 2020; Revised 8 July 2021; Accepted 21 September 2021; Published 23 October 2021

Academic Editor: Zbigniew Krzemianowski

Copyright © 2021 Can Luo et al. This is an open access article distributed under the Creative Commons Attribution License, which permits unrestricted use, distribution, and reproduction in any medium, provided the original work is properly cited.

The intake grid is always installed technically to protect the impeller at the entrance of the waterjet propulsion device's inlet duct affecting its performance. Therefore, this study discussed the complex features of the circular, rectangular, and streamlined intake grid. Consistent geometry size of the intake grid mentioned above is to be maintained to guarantee the identical flow capacity at the entrance of inlet duct. Using experimental and simulated method, the outcomes are drawn as below. Rather than the circular and rectangular intake grid, the streamlined intake grid can improve the hydraulic performance of the waterjet propulsion device. The numerical method is proved to be correct as the consistence of the hydraulic characteristic between the test and simulated results. The causes of hydraulic loss in the contraction segment and straight pipe segment are the intake grid and the inflow velocity, respectively; meanwhile, the loss in the belt pipe segment owes to the vortex, flow separation, and impact on the back. The intake grid has a positive effect on the depth of the inlet velocity profile, but a negative effect on the width of it. The intake grid installation results in thrust reduction, the progress of velocity-weighted average angle, and the regress of axial velocity uniformity. The performance of waterjet propulsion device is complex and evaluated by the hydraulic performance index (HPI), thrust performance index (TPI), and characteristic of flow pattern index (CFPI). Based on the three evaluation indexes, the streamlined scheme is raised to be the recommended scheme.

1. Introduction

The waterjet propulsion system is invented to propel the ship by utilizing the interaction force between the sea and the hull about 300 years ago. In the following development process, five sections of waterjet propulsion device, including hydraulic pump type, intermittent type, bottom plate type, external hanging type, and tail plate type, are experienced [1, 2]. Waterjet propulsion device consists of the inlet duct, pump, nozzle, steering device, and other attachments. It is often adopted in the high-speed craft due to the flexible driving, excellent maneuverability, high sailing speed, outstanding cavitation performance, and high efficiency. When

the speed extends to 25 knots, the total efficiency even extends to 60% [3].

As similar with the trash rack in the front of hydraulic structure, the intake grid is always installed at the entrance of the inlet duct to prevent the floating objects, such as water plants, swimming ocean life, and marine wastes including foam, plastic, and wood in the stream, being sucked into the waterjet pump, thus protect it [4]. Or else, blades may be damaged accidentally. However, the intake grid will cause enhancing hydraulic loss, lower efficiency, and thrust deduction. Hydraulic loss in the inlet duct mainly consists of local loss especially at the elbow, dissipation loss at the entrance, and the surface friction loss, nearly 7%–9% shaft

power is consumed [5]. If intake grid is applied, interaction between the intake grid and inlet duct may result in severe energy dissipation. Concentrating on the performance in the waterjet propulsion device, especially the intake system, efforts are taken on. Duerr et al. [6] numerically simulated the velocity fields entering the pump of a waterjet-propelled ship with the hull to evaluate the level of nonuniformity in axial velocity at various shaft and ship speeds. Pressure fluctuations on the impeller blade and the virtual stream tube were obtained using steady-state MFR method and fully transient moving mesh method [7]. Park et al. [8] performed the secondary flow in the inlet duct, the recovery of axial flow, and tip vortex and predict the performances of thrust and torque. Huang et al. [9] multiobjectively optimized the inlet duct with the modified Non-dominated Sorting Genetic Algorithm-II (NSGA-II). Fujisawa [10] experimentally evaluated the system and the unit performances of waterjet propulsion systems in a water tunnel and also measured the losses in the intake and the nozzle. Xu and Zhou [11] numerically studied the effects of inflow directions (forward, backward, and oblique motion) on the flow field within a flush type of inlet duct of waterjet. Cao et al. [12] investigated the performance deviation between the uniform and non-uniform suction flows and explain its generating mechanism with numerical method. Miorini et al. [13] studied the instantaneous and phase-averaged inner structures of the tip flow and evolution of the tip leakage vortex (TLV) in the rotor of an axial waterjet pump. Gong et al. [14] numerically simulated the internal flow field of a waterjet-propelled ship and evaluated the unsteady hydrodynamic performance of the impeller. Xu et al. [15] investigated the rotating stall and proposed the groove to suppress this phenomenon effectively. Luo et al. [16] applied the detached eddy simulation (DES) turbulence model and the Zwart-Gerber-Belamri (ZGB) cavitation model to simulate the cavitation flow in the waterjet pump under the critical working conditions when the performance drops sharply. Moreover, the intake grid is also focused on. Zhou et al. [17, 18] established the motion differential equations of single degree of freedom and multidegree of freedom systems with theoretical analysis, finite element calculation, and experimental research. The cause of the grid damage is concluded with the strength and modal calculations on the intake grid using commercial code MSC. Chang et al. [19] analyzed the interaction between each intake grid with computational fluid dynamics method and obtained that the static strength is not the reason why the intake grid fractured repeatedly. Luo et al. [20] used the Galerkin method to numerically solve the vortex-induced resonance equation of the grid structure in uniform flow field and analyzed the causes of grid fracture damage. Wang et al. [21] figured out that installing an intake grid will result in insufficient water supply capacity, which leads to the decreased navigation speed below the designed value. Wang et al. [22, 23] represented that setting an intake grid will increase the flow loss, reduce the propulsion performance of the ship, cause the flow at the inlet to decrease, the IVR (Inlet Velocity Ratio) becomes smaller, and the efficiency of the inlet runner decreases. To improve the propulsion performance of the vessel, the inlet is changed from elliptical shape

to “semielliptical + rectangle” shape, the outlet diameter of the inlet duct passage is decreased, and the angle between the intake grid and the horizontal plane is reduced. 21st ITTC presented the definition of the capture area and stream tube covered by the hull and the virtual face where ocean water passing through into the device [24]. All the above, CFD has become a mature research method and has been applied in the rotating machinery. In this study, the performance of waterjet propulsion device with the intake grid is discussed using CFD software and experimental test.

2. Model Experiment

2.1. Test Loop System. As shown in Figure 1, closed test loop system includes two circulation cycles, which are the small-sized water tunnel and the secondary loop. In the small-sized water tunnel, centrifugal auxiliary pump provides navigation speed. By adjusting the butterfly valve, various navigation speeds are provided. The electromagnetic flow meter is used to measure the discharge. The expansion joint is applied to solve the installed location deviation of each facility at the below. Steady flow circumstance away from the capture area is expected by applying rectifying device at the upstream and downstream. When the water tunnel is operating, water begins to flow and form the navigation speed. By starting the variable-frequency motor, the power is conducted to the mixed propulsion pump through the shaft and the coupling connected with stainless flanges, screws, and flexible solids, then the blades begins to rotate. The sucked water passing through the inlet duct flows back into the small-sized water tunnel. In the secondary loop, electric butterfly valve is used to control the discharge accurately.

2.2. Test Facilities. The flow rate, head, revolution, and torque information is collected to trace the hydraulic performance curve. The flow rate in the small-sized water tunnel, and the secondary loop are measured by electromagnetic flow meters. The head is calculated by the static pressure difference between the upstream and downstream. The static pressure is measured by the pitot pipes connected with the pressure transmitter. The pitot pipe at the upstream side is located more than $1D$ away from the entrance of the inlet duct, whereas the pitot pipe at the downstream side is situated more than $3D'$ away from the export of the pump. In which, D and D' are the inlet's and outlet's diameter independently. Variable-frequency motor in accordance with Danfoss inverter covering the rotating speed range of 5–50 Hz is employed to provide multiple rotating speeds. Using the torque meter and matching indicator, the torque magnitude is measured.

2.3. Uncertainty Analysis. The secondary loop or measuring test rig, static pressure value with the precision lower than $\pm 0.2\%$ is acquired. The assurance of the electromagnetic flow meter is $\pm 0.5\%$. Torque meter also can measure the torque and the shaft power in $\pm 0.2\%$.

In formula (1), the system uncertainty is calculated by the uncertainty of pressure, head, and discharge.

FIGURE 1: Sketch of test loop system. 1, transition section; 2, rectifying device; 3, inlet duct; 4, mixed propulsion pump; 5, electromagnetic flow meter; 6, butterfly valve; 7, auxiliary pump.

$$E_{\eta S} = \pm \sqrt{E_{PS}^2 + E_{HS}^2 + E_{QS}^2}, \quad (1)$$

where E_{PS} is the power uncertainty matching with torque meter, E_{HS} is the head uncertainty consistent with pressure transmitter, E_{QS} is the discharge uncertainty in accordance with electromagnetic flow meter, and $E_{\eta S}$ is the composite accumulate uncertainty due to pressure, head, and discharge contributing to the efficiency according to the following:

$$\eta = \frac{\rho g Q H}{P_{\text{shaft}}}, \quad (2)$$

$$\eta = \frac{9550 \rho g Q H}{T N}, \quad (3)$$

where Q is the volume discharge in m^3/s , H is the head in m , P_{shaft} is the shaft power in kW , T is the torque in $N \cdot m$, N is the rotating speed in rad/s , ρ is the density of water about 1000 kg/m^3 , and g is the acceleration due to gravity of 9.81 m/s^2 . η is the efficiency.

The calculated dimensionless uncertainty is $\pm 0.57\%$, meaning the experimental error of efficiency is below 0.57% .

3. Numerical Preparatory

3.1. Computational Domains. The boundary layer at the bottom of the hull will result in the uneven flow at the entrance of the inlet duct; therefore, computational domains include the waterjet propulsion device and the water body at the bottom of the hull shown in Figure 2. The waterjet propulsion device is composed of inlet duct, impeller, guide vane, and nozzle. The size of water body employed in the calculation is $30D \times 5D \times 8D$.

3.2. Mesh Generation. In general, structure mesh contributes to more superb convergence, smaller truncation error, and less calculation resource. The intake grid is not attached in the original waterjet propulsion device marked as original scheme in this study. Each subdomain is blocked and divided into structured hexahedral mesh in ANSYS ICEM, due to very different geometric feature listed in Table 1. O-type strategy is employed to generate structure hexahedral mesh of the nozzle. The guide vane and impeller adopts H-type strategy. The mesh around the blades of impeller and guide

vane, near the shaft and on the contact zone between the propulsion device and water body, is refined. The boundary layer is added, and five near-wall prism layers are adopted. The y^+ value of propulsion pump is less than 100. However, the geometrical shape of inlet duct is too complex when the intake grid is equipped. Thus, the inlet duct with intake grids is generated with unstructured meshes. Table 1 shows the three cross-sectional planes to show the detailed meshes surrounding the streamlined, rectangular, and circular intake grid, and the mesh quantity is about 4.2 million, which is four times larger than the original scheme. The section of rectangular intake grid is more regular than other schemes, different mesh strategies are executed. In other words, the meshes are encrypted near the streamlined and circular intake grid to adapt to their shapes.

The calculated result may vary too much when generating meshes of different quantities. Thus, the grid convergence index (GCI) and validation and verification (V&V) are applied to obtain the reasonable mesh, guaranteeing the accuracy of simulated results. Block structure of each subdomain is constant but the growth of nodes varies, so that different quantities of mesh is employed.

By evaluating several numerical simulations, GCI has been demonstrated as the most credible method and is widely recommended to estimate the numerical uncertainty [25]. Therefore, the research also utilizes GCI method. Three meshes are generated to perform the grid sensitivity, and the grid refinement ratio in the cartesian coordinate directions is kept as 1.3. Considering the distinct meshes of each subdomain, monitoring points are selected shown in Figure 3. Note that P1 is set on the inlet of the stream tube in the water body, P2 and P3 are located in the intake grid and elbow of the intake duct, P4 and P5 are probed at the leading edge (LE) and trailing edge (TE) along the midchord curve in the impeller, P6 and P7 are also situated at the identical edges in the guide vane, and P8 is arranged in the nozzle. Figure 4 illustrates the pressure coefficient p' growth trend along the distance from the nozzle exit x/l and GCI_{fine}^{21} value calculated according to the literature [26]. The uncertainty value is less than 3%. It is very obvious that the simulated pressure is closest to the extrapolated value when the mesh quantities of each scheme are 3.11 million, 6.32 million, 6.28 million, and 6.3million. When the mesh increases continuously, extra computing resources are occupied. Therefore,

FIGURE 2: Computational domains.

TABLE 1: Mesh of each subdomain.

Subdomain	Mesh	Quantity (10^5)
Nozzle		0.6
Guide vane		5.5
Impeller		8.9
Inlet duct		10.4

TABLE 1: Continued.

Subdomain	Mesh	Quantity (10 ⁵)
Water body		5.7
	42.7
	42.1
Intake grid		42.5

FIGURE 3: Location details of monitoring points.

(a)

(b)

FIGURE 4: Continued.

FIGURE 4: Grid convergence index analysis of each scheme (a) Original scheme. (b) Circular scheme. (c) Rectangular scheme. (d) Streamlined scheme.

the mesh of each scheme above are selected to accomplish the follow-up research.

3.3. Validation and Verification. The computational fluid dynamics results are recommended to be validated and verified in many fields; AIAA and ITTC presented the guide for the CFD validation and verification [27, 28]. Zhang et al. [29, 30], Kandasamy et al. [31] and Huang et al. [32] covered the CFD validation and verification to convince the mesh and method related to the waterjet ships and device system performance. Therefore, this study finishes the CFD validation and verification using ITTC guide. The simulation numerical uncertainty U_{SN} is calculated by iteration number uncertainty U_I , grid size uncertainty U_G , time step uncertainty U_T and other

parameters uncertainty U_P , and the expression is given in formula (4).

This research is a steady simulation and adopts the CFD commercial code without revising the simulation model. In other words, time step uncertainty U_T and other parameters uncertainty U_P do not contribute to simulation numerical uncertainty U_{SN} . However, previous works has demonstrated that the iteration number uncertainty U_I is ignorable compared with the grid size uncertainty U_G . Finally, the simulation numerical uncertainty U_{SN} is simplified and approximately equals to grid size uncertainty U_G . Therefore, the mesh independency analysis is quite necessary.

$$U_{SN} = \sqrt{U_I^2 + U_G^2 + U_T^2 + U_P^2}. \quad (4)$$

Five meshes of each scheme are generated to comply the mesh independency analysis. Divided by the head and efficiency of the fourth mesh for each scheme, the head and efficiency is normalized. When the meshes of each scheme exceed to 3.11 million, 6.32 million, 6.28 million, and 6.34 million as shown in Figure 5, the diversification of the normalized head and efficiency is less than 0.5%.

3.4. Boundary Conditions. In ANSYS CFX solver, boundary conditions are set up, and the finite volume method is integrated. Then, the steady flow model is applied in this simulation. In the computational domains, two cartesian coordinates are adopted in this investigation according to the stationary and rotating references. The rotating zone is the impeller whose revolution speed is 700 rpm; the others are stationary zone. The inlet boundary at the entrance of the water body below the hull is set as navigation speed 8 m/s. The water body outlet is applied as outflow boundary condition, and the nozzle outlet is applied as mass flow. The connection between each subdomain is set as interface. In which, the “impeller-inlet duct” interface and “impeller-guide vane” interface are frozen rotor, while the others are stationary interfaces. Because the flow in this study is incompressible, Reynolds average N-S equation and continuity equation are utilized to describe this flow. The standard k- ϵ turbulence model is applied, and the upwind scheme is the first-order upwind scheme. The convergence accuracy is 10^{-5} .

4. Test Validation and Verification

Test validation is essential to verify the numerical simulation method. The hydraulic performance curve includes flow rate-head curve (Q' - H' curve) and flow rate-efficiency curve (Q' - η' curve). The flow rate, head, shaft power, and revolution speed are measured, yet the efficiency is calculated using formula (2) or (3). The impeller rotates at 400 rpm, meanwhile the navigation speed is 0.45 m/s. Normalized physical quantities are obtained by being divided by the flow rate, head, and efficiency at the best efficiency point and compared with test results. The results are given in the literature [16]. The trends of the Q' - H' curve and Q' - η' curve are consistent (Figure 6). In which, the simulated result is in good agreement with the test result. Hence, the numerical simulation method is reliable and will be adopted in the following research.

5. Result Analysis

5.1. Research Scheme. Rectangular, circular, and streamlined intake grids are attached to the waterjet propulsion device shown in Figure 7. W_{1s} , W_{1r} , and W_{1c} are the gap $0.14D$ away from the neighboring intake grid. L_s and L_r are the length of the streamlined and rectangular intake grid, valuing $0.055D$. D_c is the diameter of the circular intake grid, equaling to L_s and L_r . W_{2s} and W_{2r} are the width of the streamlined and rectangular intake grid, sharing the same size $0.008D$. In the other words, this study guarantees the inlet flow capacity of waterjet propulsion pump by

controlling the geometry size of circular, rectangular, and streamlined intake grids on above. For each scheme, six symmetrical grids are applied, and the distance between each grid is constant.

5.2. Characteristic Index. The characteristic indexes contain the hydraulic performance index (HPI), the thrust performance index (TPI), and the characteristic of flow pattern index (CFPI). In which, the hydraulic performance index (HPI) consists of head and efficiency, and the thrust performance index (TPI) is the thrust per shaft power F_{P-T} . The characteristic of flow pattern index (CFPI) includes axial velocity uniformity V_{au} and velocity-weighted average angle θ_a obtained from the following:

$$F_P = \rho A_{\text{nozzle}} v_{\text{out}} (v_{\text{out}} - \alpha v_s), \quad (5)$$

$$F_{P-T} = \frac{F_P}{P_{\text{shaft}}}, \quad (6)$$

$$V_{au} = \left[1 - \sqrt{\sum_{i=1}^n \left(\frac{v_{ai}/v_a - 1}{n} \right)^2} \right] \times 100\%, \quad (7)$$

$$\theta_a = \frac{\sum_{i=1}^n [(90^\circ - \arctan(v_{ti}/v_{ai}))v_{ai}]}{\sum_{i=1}^n v_{ai}}, \quad (8)$$

where F_P is the thrust in N; P_{shaft} is the shaft power in kW; F_{P-T} is the thrust per shaft power in N/kW; A_{nozzle} is the area of the nozzle outlet in m^2 ; v_{out} is the velocity of the nozzle outlet in m/s; v_s is the navigation speed in m/s; α is the boundary layer influence coefficient [1]; ρ is the density of water about 1000 kg/m^3 ; v_a is the average axial velocity on the outlet section of the inlet duct in m/s; n is the node; and v_{ai} is the axial velocity of each node on the outlet section of the inlet duct in m/s. v_{ti} is the tangential velocity of each node on the outlet section of the inlet duct in m/s.

5.3. Hydraulic Performance and Loss

5.3.1. Hydraulic Characteristics. The hydraulic characteristics consists of head and efficiency. Divided by the head and efficiency of the best efficiency point for the original scheme, dimensionless head and efficiency of each scheme is calculated. Dimensionless head and efficiency of each scheme are plotted in Figure 8(a) and Figure 8(b).

In Figure 8(a), the head increases noncontinuously as the enhancing flow rate for all schemes. When the flow rate is $0.5Q'$, which is valley point, obvious reduction happens; the head of the original scheme is highest, followed by the rectangular scheme and the streamlined scheme, then is the circular scheme. When the flow rate exceeds $0.83Q'$, the Q' - H' performance of streamlined scheme is slightly higher than other schemes.

In Figure 8(b), when the flow rate is $0.33Q'$, which is the small flow rate, the efficiency of each scheme is basically consistent; when the flow rate is $0.5Q'$, $0.67Q'$, and $0.83Q'$, the efficiency of original scheme is larger than the other schemes,

FIGURE 5: Continued.

FIGURE 5: Mesh independency column: red, blue, and purple column are the designed flow rate condition; 1.17 and 1.33 times are best efficiency point condition (Q'), respectively. (a) Original scheme. (b) Circular scheme. (c) Rectangular scheme. (d) Streamlined scheme.

FIGURE 6: CFD and Test results (Luo et al.). (a) Q' - H' curve. (b) Q' - η' curve.

the disparity between the original and the streamlined scheme decreases gradually; when the flow rate exceeds to $0.83Q'$ especially large flow rate condition, the efficiency of streamlined scheme is obviously higher than the others, and this trend becomes more obvious as the flow rate increases, even when the flow rate $1.33Q'$, the efficiency of streamlined scheme is approximately two times than the original scheme, three times than the rectangular scheme, and four times than the circular scheme. When the flow rate exceeds $0.83Q'$, the efficiency of the streamlined scheme is higher than the original scheme.

Thus, the installation of the streamlined intake grid has active effect on the hydraulic performance of the waterjet

propulsion device. On the opposite, the efficiency decreases obviously when the circular and rectangular intake grid is installed.

5.3.2. Hydraulic Loss. The inlet duct generally consists of contraction segment, belt pipe segment, and straight pipe segment. The hydraulic loss proportions of each part are drawn in Figures 9–11, in which, the data of original scheme stem from the literature [33].

Figure 9 shows the trend of the proportion of hydraulic loss in the contraction segment. On the whole, the hydraulic

FIGURE 7: Intake grid schemes and geometry sizes. (a) Intake grid schemes. (b) Streamlined intake grid. (c) Rectangular intake grid. (d) Circular intake grid.

loss in the contraction segment of each scheme drops. Under all operating conditions, the hydraulic loss proportion of the original scheme is less than that of three grid schemes. More than that, the rectangular intake grid scheme is greater than other intake grid schemes, followed by streamlined and circular intake grid. With the increasing flow rate, this trend tends to be more obvious. The intake hydraulic loss, the contraction hydraulic loss, and the intake grid hydraulic loss are recognized in the contraction segment. First of all, intake hydraulic loss. It depends on two aspects: intake geometry and inlet velocity. The geometric characteristics of intake are the inlet shape, length-width ratio, and inflow angle. Considering that the intake shape and pitch angle of the inlet duct of the four schemes are identical, the inflow velocity of

each scheme is slightly different, but the maximum difference is within 1.2%. Therefore, the intake loss is not the reason for the difference of hydraulic loss in contraction segment. Then, the contraction hydraulic loss mainly depends on the inflow angle, contraction coefficient, lip angle, and inflow velocity. The four schemes have the consistent parameter, so the contraction loss is also not the reason for the divergence of hydraulic loss in the contraction segment. At Last, the intake grid hydraulic loss. This part of hydraulic loss is related to the shape of the intake grid, the length-width ratio, the grid clearance, the inflow angle, and the inflow velocity. As configured, the length-width ratio and the grid clearance of each scheme are coincident. Moreover, the discrepancy of the inflow angle can be ignored as shown

FIGURE 8: Hydraulic performance of each scheme. (a) $Q'-H'$ performance. (b) $Q'-\eta'$ performance.

in Figures 12–15 and so is the inflow velocity. It should be observed that the sole discrimination is the consistence of the intake grid's shape. Thus, the shape of the intake grid absolutely contributes to the intake grid loss. Previous study [3], which is not involved the circular intake grid, pointed out that the cross-sectional coefficient of intake grid is directly related to the shape of intake grid, and the rectangular intake grid holds the largest cross-sectional coefficient. On the contrary, the streamlined intake grid is opposite. It is completely consistent with the trend obtained in Figure 9.

Figure 10 shows the hydraulic loss comparison of belt pipe segment. It is found that the hydraulic loss of belt pipe segment in the original scheme is the largest, followed by the circular intake grid scheme. Except for $0.33Q'$, the hydraulic loss of rectangular intake grid scheme is greater than that of streamlined intake grid scheme when the flow rate is less than the optimal flow rate. But when the flow rate is greater

than the optimal flow rate, the trend is opposite. With the increasing flow rate, the loss proportion trend of the belt pipe segment in each scheme is completely consistent, that is, it is increasing. In Figures 12, 13, and 15, symmetrical critical vortices are surveyed on the both sides at the back of the belt pipe segment in the original scheme under the small flow rate condition, while slight flow separation occurs in the circular and streamlined intake grid schemes, which indicates that the dominant hydraulic loss source of the belt pipe segment under the small flow condition is mainly vortex and flow separation. Because the geometry of the belt pipe segment in each scheme are identical and the inflow velocity is not much diverse, the cornering velocity increases with the raising flow rate. It implies that the drastic impact on the pump shaft and the side wall of the belt pipe segment tends to be more apparent. In other words, the hydraulic loss of the belt pipe segment mainly comes from the water impact at

FIGURE 9: Hydraulic loss comparison of contraction segment.

FIGURE 10: Hydraulic loss comparison of belt pipe segment.

this time. In a nutshell, the hydraulic loss in the belt pipe segment is mainly activated by vortex, flow separation, and impact, and the main influencing factors include the geometry of the belt pipe segment and the inflow velocity.

Figure 12 shows the trend of the proportion of hydraulic loss in the straight pipe segment of each scheme. The hydraulic loss of each scheme increases with the raising flow rate. Under the constant flow rate, the proportion of hydraulic loss in the original scheme grows, which is obviously higher than that of other schemes, followed by circular, streamlined, and rectangular intake grid schemes. For the straight pipe segment, the hydraulic loss is mainly the hydraulic loss along the way. Considering the consistent flow rate and input energy of each scheme, massive energy consumed in the contraction segment of each intake grid scheme and the completely identical geometry of the straight segment of each scheme, it is not difficult to infer that the

velocity of the original scheme is significantly higher than that of other schemes. That is the reason why the trend of hydraulic loss of straight pipe segment of each scheme happens just in the way above.

5.4. Stream Tube and Thrust Performance. The work principles of screw propeller and waterjet propulsion system completely varies. Therefore, the methods to calculate the thrust are also different, in which, the thrust is measured by the axial force on the shaft of screw propeller. On the contrary, the thrust is calculated by adopting the momentum method proposed by ITTC. The capture area consists of the hull, the stream tube, the inlet duct, the propulsion pump, and nozzle and then is put forward. The outline of the hull, the inlet duct, the propulsion pump, and nozzle is actual, but the stream tube is defined as the flow passed through the

FIGURE 11: Hydraulic loss comparison of straight pipe segment.

FIGURE 12: Side view of stream tube for original scheme. (a) Small flow rate (0.33 Q'). (b) Best efficiency point (Q'). (c) Large flow rate (1.33 Q').

device and drawn virtually. According the definition of 21st ITTC Waterjet Committee (1996), the distance between the inlet velocity profile of the stream tube and the inlet duct is one impeller diameter. The thrust is obtained by the momentum discrepancy on the inlet and outlet of the stream tube. Therefore, the flow characteristics of stream tube should be analyzed.

5.4.1. Stream Tube Characteristics. Figure 12 is the side view of stream tube for original scheme at different working conditions, in which, the subfigure on the left is the flow passing by the entrance of the inlet duct, while the subfigure on the right is the stream tube given in the literature [34]. In the left subfigure, the black line is the virtual dividing boundary of the sucking stream tube by being redrawn according the right subfigure at each operating conditions.

Absolutely, the entry of inlet duct should be distinguished from the inlet of stream tube. Under the small flow rate, inverse flow exists at the back of inlet duct, and the stream is separated into two directions at the entrance of inlet duct. The shape of stream tube tends to the outflow direction of the inlet duct. To distinguish from each other, the dividing boundary (DB) is drawn with a black line virtually, and the top point of DB is marked as black star point. According to stream tube in the right subfigure, the area above the DB is the stream tube. On the contrary, a few stream still flows into the inlet duct and then is pushed out from the duct, finally flows toward to the stern. The intersection point between the DB and inlet duct is defined as stagnation point, which is signed as a red pentagram. When the discharge becomes larger, less stream flows out of the device, the location of the stagnation point gets nearer to the lip of inlet duct. Moreover, the top point of DB also shows the same trend. As

FIGURE 13: Side view of stream tube for circular scheme. (a) Small flow rate ($0.33Q'$). (b) Best efficiency point (Q'). (c) Large flow rate ($1.33Q'$).

FIGURE 14: Side view of stream tube for rectangular scheme. (a) Small flow rate ($0.33Q'$). (b) Best efficiency point (Q'). (c) Large flow rate ($1.33Q'$).

the result, the top point of DB and the stagnation point coincides practically.

Figures 13–15 are the side view of stream tube for each scheme at different working conditions. The DB shows the same trend at the constant working conditions. At the small flow rate, the inverse flow on the back of the inlet duct is well improved. It means the stream is rectified when it flows passing the intake grid. Furthermore, the rectification result is related with the topologies of the intake grid. In addition, the stream tube DB outlines of the other schemes are completely consistent with the original scheme. Yet, the top point of DB moves to the lip, and the stagnation point is above the crossover point between the intake grid and inlet duct. Minority stream is pumped into the device through the intake grid, on the contrary, the majority stream is inhaled and then pressed out from it. With the increasing flow rate,

this phenomenon disappears, the out-pressed flow diminishes, all the flow passing by the intake grid is entirely sucked into the device. The tendency of the stagnation point and the top point of DB is matched with the original scheme, when the intake grid is attached.

Figures 16–18 show the shape of the inlet velocity profile of the stream tube under different flow conditions. In the figures 1–7 are the inlet velocity profile under the flow conditions of $0.33Q'$, $0.5Q'$, $0.67Q'$, $0.83Q'$, Q' , $1.17Q'$, and $1.33Q'$, respectively. In most working conditions, the inlet velocity profile of each intake grid scheme is not semi-elliptical, and the outline of the inlet velocity profile in the center tends to the water surface, whereas the circular and streamlined intake grids are semielliptical under large flow conditions. On the whole, with the increase of the flow rate, the width and depth of the inlet velocity profile increase, and

FIGURE 15: Side view of stream tube for streamlined scheme. (a) Small flow rate ($0.33Q'$). (b) Best efficiency point (Q'). (c) Large flow rate ($1.33Q'$).

FIGURE 16: Inlet velocity profile of the stream tube of circular intake grid scheme.

FIGURE 17: Inlet velocity profile of the stream tube of rectangular intake grid scheme.

the area gradually increases. The geometrical characteristics of the inlet velocity profile of the circular and the streamlined intake grid are not too discrepant. On the contrast, which of rectangular intake grid is opposite. It is indicated that the section shape of the intake grid is one of the factors influencing the geometrical characteristics of the inlet velocity profile.

Figure 19 shows the comparison of inlet velocity profiles of different grid schemes under the valley point of rotating stall region, the initial point of rotating stall region and the large flow condition [31]. In the figure, the lines colored with red, green, yellow, and cyan are the outlines of the inlet velocity profile of each scheme, respectively. At the valley point, the inlet velocity profile of the original scheme is obviously wider and deeper

FIGURE 18: Inlet velocity profile of the stream tube of streamlined intake grid scheme.

(a)

(b)

(c)

FIGURE 19: Comparison of the inlet velocity profile of each intake grid scheme. (a) Valley point of rotating stall zone. (b) Initial point of rotating stall zone (c) Large flow rate condition point.

FIGURE 20: Geometrical parameters of the inlet velocity profile. (a) Area of the inlet velocity profile. (b) Depth of the inlet velocity profile. (c) Width of the inlet velocity profile.

than the other schemes. The width and depth of the inlet velocity profile of the circular intake grid and the streamlined intake grid basically coincide with the original scheme; however, the rectangular intake grid are different. At the initial point, the inlet velocity profile of the stream tube in the original scheme is the widest and deepest, followed by the streamlined and the circular intake grid scheme. The rectangular intake grid scheme shows the opposite trend, while the distinction between the streamlined intake grid and the original scheme is almost negligible. Under large flow conditions, the inlet velocity profiles of the original, the circular, and the streamlined intake grid schemes are all semielliptical, and the outline is basically coincident; nevertheless, the central part of the inlet velocity

profile of the rectangular intake grid scheme still tends to the water surface.

Based on the depth, width, and area of the original scheme under the optimal conditions, the stream tube's parameters of each scheme are normalized and plotted in Figure 20. The area, depth, and width of the stream tube continue to increase with the increasing flow rate. Under the flow condition of $0.33Q'$, the geometric parameters of the inlet velocity profile of the circular and streamlined intake grid are slightly different from the original scheme, and the rectangular intake grid has a large difference. With the enlarging flow rate, the geometric parameters of the inlet velocity profile of the streamlined intake grid scheme are

FIGURE 21: Dimensionless thrust per shaft power of each scheme.

FIGURE 22: Analysis sections.

FIGURE 23: Axial velocity uniformity and velocity-weighted average angle on the outlet of inlet duct.

closest to the original scheme, followed by the circular intake grid scheme, and finally the rectangular intake grid scheme.

5.4.2. Thrust Performance. Based on the momentum method, the stream tube performance is concluded and the thrust is calculated. Then, the thrust per shaft power of each working condition is obtained with formulas (7) and (8). Finally,

dimensionless thrust per shaft power of each scheme is calculated by dividing the thrust per shaft power of the best efficiency point for the original scheme and recorded as F'_{P-T} . The thrust per shaft power of the original scheme is the largest, followed by the rectangular scheme, the streamlined scheme, and the circular scheme, as shown in Figure 21, in which, the value of the rectangular scheme and the streamlined scheme is nearly the same.

FIGURE 24: Radar charts of each scheme.

5.5. Axial Velocity Distribution. The uniformity on the outlet of the inlet duct will result in the unbalanced force on the impeller. Therefore, the uniform feature on the outlet should be drawn and is sliced in Figure 22. The flow direction of the inlet passage is parallel to the axial direction (+Z axis).

The axial velocity uniformity (V_{au}) and the velocity-weighted average angle (θ_a) of each scheme on the section are calculated and analyzed with formulas (7) and (8). Figure 23 illustrates the dimensionless axial velocity uniformity and the velocity-weighted average angle marked as V_{au}' and θ_a' based on the original scheme at best efficiency point. The results show that when the intake grid is installed, the axial velocity uniformity on the outlet of inlet duct decreases. Then, the rectangular scheme has the largest drop, followed by the streamlined scheme and the circular scheme while the velocity-weighted average angle on the outlet of inlet duct increases obviously. The flowing smoothness on the outlet of inlet duct has been improved. Then, the streamlined scheme enhances most, then the circular scheme, the rectangular scheme is the smallest.

5.6. Comprehensive Confrontation. According to the data under best efficiency point of original scheme, the dimensionless hydraulic performance index (HPI), the thrust performance index (TPI), the characteristic of flow pattern index (CFPI), and high efficiency zone of each scheme are obtained. Relative efficiency η' , axial velocity uniformity V_{au}' , velocity-weighted average angle θ_a' , and thrust per shaft power F_{p-T}' are listed in Figure 24. The performance of the streamlined scheme is slightly better than the other two schemes.

6. Conclusion

This study establishes the computational domain consisting of water body, intake grid (circular, rectangular,

and streamlined), inlet duct, waterjet propulsion pump, and nozzle to investigate the performance of the waterjet propulsion device with intake grid numerically. Then, a waterjet propulsion test loop system is built up to test the performance. The following conclusions can be drawn:

- (1) The streamlined intake grid has active effect on the hydraulic performance of the waterjet propulsion device. On the opposite, the efficiency decreases obviously for the circular and rectangular intake grid.
- (2) By test validation, the flow rate-head curve and flow rate-efficiency curve of test and simulated results are consistent and significantly matched. That is, the numerical simulation method is credible.
- (3) The loss in the inlet duct is sum in the contraction segment, belt pipe segment, and straight pipe segment. But their causes are divergent completely. In detail, firstly, the intake grid is the major element resulting into the hydraulic loss in the contraction segment. Secondly, the vortex, flow separation, and impact are the governing factors contributing to the hydraulic loss in the belt pipe segment. Lastly, the inflow velocity is the key reason of the hydraulic loss in the straight pipe segment.
- (4) The geometrical parameters of the outer contour are affected by the installation of intake grid. The inlet velocity profile of original scheme is wider than that of intake grid schemes, but the depth of the inlet velocity profile shows the adverse trend. The top point of dividing boundary and the stagnation point will coincide as the increasing the flow rate. The thrust decreases when the intake grid is attached.
- (5) The intake grid results in the improvement of velocity-weighted average angle and the suppression of axial velocity uniformity.
- (6) To comprehensively compare the hydraulic performance and loss, the stream tube and thrust performance and internal flow characteristics of the waterjet propulsion system with intake grid, complex evaluation system consisting of hydraulic performance index, thrust performance index, and characteristic of flow pattern index marked as HPI, TPI, and CFPI are established. Above all, the overall performance of streamlined scheme is better than other schemes and promoted as the recommended scheme.

Data Availability

The data used to support the findings of this study are available from the corresponding author upon request.

Conflicts of Interest

The authors declare that there are no conflicts of interest regarding the publication of this paper.

Acknowledgments

This research was funded by Jiangsu Province Science Foundation for Youths (Grant no. BK20170507), Natural Science Foundation of the Jiangsu Higher Education Institutions (Grant no. 17KJD580003), Jiangsu Planned Projects for Postdoctoral Research Funds (Grant no. 1701189B), Open Research Subject of Key Laboratory of Fluid and Power Machinery (Xihua University), Ministry of Education (Grant nos. szjj2019-018), Science and Technology Innovation and Cultivation Fund of Yangzhou University (Grant no. 2019CXJ076), National Natural Science Foundation of China (Grant nos. 51779214 and 51806187), Priority Academic Program Development of Jiangsu Higher Education Institutions (PAPD), Jiangsu Province 333 High Level Talents Training Project (Grant No. (2018) III-1827), Peak Plan Six Talents in Jiangsu Province, and Key Project of Water Conservancy in Jiangsu Province (Grant no. 2018042), Key Laboratory of Modern Agricultural Equipment and Technology (Jiangsu University), and Ministry of Education (No. NZ201604).

References

- [1] F. P. Tang, *Design and Turbulence Numerical Analysis of Waterjet Axial-Flow Pump*, Shanghai Jiao Tong University, Shanghai, China, 2006.
- [2] J. L. Allison, "Marine waterjet propulsion," *Transaction of SNAME*, vol. 101, pp. 275–335, 1993.
- [3] P. Z. Jin, *Marine Water-Jet Propulsion*, National Defence Industry Press, Beijing, China, 1st edition, 1986.
- [4] State Oceanic Administration, "Bulletin of China marine ecological environment status," <http://www.soa.gov.cn/zwgk/hygb/zghyhjzlgbl/>.
- [5] J. S. Carlton, *Marine Propellers and Propulsion*, Butterworth-Heinemann Ltd, Oxford, UK, 3rd edition, 2012.
- [6] P. Duerr, K. D. von Ellenrieder, and von Ellenrieder, "Scaling and numerical analysis of nonuniform waterjet pump inflows," *IEEE Journal of Oceanic Engineering*, vol. 40, no. 3, pp. 701–709, 2015.
- [7] N. W. H. Bulten and R. Verbeek, "CFD simulation of the flow through a water jet installation," in *Proceedings of the International Conference on Waterjet Propulsion 4*, RINA, Amsterdam, Netherlands, October 1998.
- [8] W. G. Park, J. H. Jang, H. H. Chun, and C. M. Kim, "Numerical flow and performance analysis of waterjet propulsion system," *Ocean Engineering*, vol. 32, no. 14–15, pp. 1740–1761, 2005.
- [9] R. Huang, Y. Dai, X. Luo, Y. Wang, and C. Huang, "Multi-objective optimization of the flush-type intake duct for a waterjet propulsion system," *Ocean Engineering*, vol. 187, Article ID 106172, 2019.
- [10] N. Fujisawa, "Measurements of basic performances for waterjet propulsion systems in water tunnel," *International Journal of Rotating Machinery*, vol. 2, no. 1, pp. 43–50, 1995.
- [11] H. L. Xu and Z. J. Zhou, "Numerical study on the effects of inflow directions on the flow field in a waterjet duct," *Chinese journal of hydrodynamics*, vol. 33, no. 2, pp. 181–186, 2018.
- [12] P. Cao, Y. Wang, C. Kang, G. Li, and X. Zhang, "Investigation of the role of non-uniform suction flow in the performance of water-jet pump," *Ocean Engineering*, vol. 140, pp. 258–269, 2017.
- [13] R. L. Miorini, H. X. Wu, and J. Katz, "The internal structure of the tip leakage vortex within the rotor of an axial waterjet pump," *Journal of Turbomachinery-Transactions of the ASME*, vol. 134, no. 3, Article ID 031018, 2012.
- [14] J. Gong, C.-Y. Guo, T.-C. Wu, and K.-W. Song, "Numerical study on the unsteady hydrodynamic performance of a waterjet impeller," *Journal of Coastal Research*, vol. 341, no. 1, pp. 151–163, 2018.
- [15] S. Xu, X. P. Long, B. Ji, and G. Li, "Investigation on the mechanism between vortex and cavitation in an axial waterjet pump," *Journal of Harbin Engineering University*, vol. 41, no. 7, pp. 951–957, 2020.
- [16] C. Luo, D. Zhang, C. Zhang et al., "Investigation of rotating stall phenomenon and optimization in mixed-flow waterjet propulsion device," *Advances in Materials Science and Engineering*, vol. 2020, no. 3, 17 pages, Article ID 9421809, 2020.
- [17] C. Zhou, *The Calculation and Analysis for Strength and Mode of Intake Grid and the Visual Program of the Gross Longitudinal Strength*, Shanghai Jiao Tong University, Shanghai, China, 2007.
- [18] C. Zhou and S. L. Zhang, "The finite element calculation and analysis for strength and mode of intake grid," *China offshore form*, vol. 22, no. 3, pp. 18–21+26, 2007.
- [19] S. P. Chang, Y. S. Wang, Z. Y. Pang, and J. M. Ding, "The hydrodynamic force analysis and strength calculation of waterjet intake grid," *Ship science and technology*, vol. 32, no. 1, pp. 110–114, 2010.
- [20] Z. Luo, Z. J. Chen, C. S. Sun, and H. F. Ai, "Vortex-induced effect and structural strength of the channel-grating in waterjet propulsion," *Ship engineering*, vol. 29, no. 2, pp. 19–22, 2007.
- [21] Y. S. Wang, S. Z. Wang, S. B. Jin, and S. Chang, "Flow capacity of an inlet for intake ducts with grilles of marine waterjet," *Journal of Huazhong University of Science and Technology (Nature Science Edition)*, vol. 40, no. 10, pp. 72–77, 2012.
- [22] S. Wang, Y. S. Wang, S. B. Jin, and J. Ding, "Effect of inlet grid on duct flow performance and waterjet propulsion performance," *Journal of Mechanical Engineering*, vol. 49, no. 14, pp. 164–169, 2013.
- [23] S. Z. Wang, Y. S. Wang, and J. M. Ding, "Study on the optimization design of duct with grid of waterjet propulsion," *Journal of Ship Mechanics*, vol. 18, no. 4, pp. 357–362, 2014.
- [24] Waterjets group, "Final report and recommendations to the 21st ITTC," in *Proceedings of the 21st International Towing Tank Conference*. Bergen and Trondheim, Norwegian, pp. 189–209, Trondheim, Norway, September 1996.
- [25] P. J. Roache, "A method for uniform reporting of grid refinement studies," *ASME-publications-fed*, vol. 158, p. 109, 1993.
- [26] I. B. Celik, U. Ghia, P. J. Roache, J. Freitas Christopher, H. Coleman, and Raad Peter, "Procedure for estimation and reporting of uncertainty due to discretization in CFD applications," *Journal of Fluids Engineering*, vol. 130, no. 7, Article ID 078001, 2008.
- [27] American Institute of Aeronautics and Astronautics, "Guide for the verification and validation of computational fluid dynamics simulations," American Institute of Aeronautics and Astronautics, Reston, VA, USA, AIAA-G-077-1998, 1998.
- [28] Ittc-Recommended Procedures and Guidelines, "Uncertainty analysis in cfd verification and validation methodology and procedures," Ittc-Recommended Procedures and Guidelines, Wuxi, China, 7.5-03-01-01, 2017.
- [29] L. Zhang, J. N. Zhang, and Y. C. Shang, "Stern flap-waterjet-hull interactions and mechanism: a case of waterjet-propelled

- trimaran with stern flap,” *Journal of Offshore Mechanics and Arctic Engineering*, vol. 142, no. 2, Article ID 021203, 2020.
- [30] L. Zhang, J. Zhang, and Y. Shang, “A practical direct URANS CFD approach for the speed loss and propulsion performance evaluation in short-crested irregular head waves,” *Ocean Engineering*, vol. 219, Article ID 108287, 2021.
- [31] M. Kandasamy, D. Peri, Y. Tahara et al., “Simulation based design optimization of waterjet propelled Delft catamaran,” *International Shipbuilding Progress*, vol. 60, pp. 277–308, 2013.
- [32] R. Huang, W. Ye, Y. Dai et al., “Investigations into the unsteady internal flow characteristics for a waterjet propulsion system at different cruising speeds,” *Ocean Engineering*, vol. 203, Article ID 107218, 2020.
- [33] C. Luo, S. Lei, L. Cheng, and H. Liu, “Research on the flow feature and energy performance in the intake passage of waterjet propulsion device,” *Ship Science and Technology*, vol. 42, no. 9, pp. 82–86, 2020.
- [34] C. Luo, L. Cheng, C. Liu, and Y. Shang, “Numerical simulation of performance of marine waterjet propulsion system,” *Journal of Drainage and Irrigation Machinery Engineering*, vol. 33, no. 5, pp. 374–379, 2015.