OPTIMIZATION OF CAVITATING FLOW CHARACTERISTICS ON RBSS OF WATERJET PUMPS

Yan, H.^{***}; Li, Q.^{*}; Zhang, Y.^{*}; Shi, H. X.^{*} & Vnenkovskaia, V.^{***}

^{*} School of Mechanical Engineering, Hefei University of Technology, Tunxi Road No. 193, Hefei 230009, Anhui Province, China

^{**} Hefei Huasheng Pumps & Valves Co., Ltd., Jinhuai Road 06, Hefei 230009, Anhui, China

^{***} Research and Production Centre "ANOD", Svobody Street, 63, Nizhniy Novgorod 603003, Russia
E-Mail: yanying0708@126.com

Abstract

Cavitation, as a key factor that influences safe and stable system operations, has been a scientific research challenge in the field of hydraulic machineries. Waterjet pumps are typical hydraulic machineries in which cavitation is generated near rotor blades suction surface (RBSS), and their geometric structures are the main factors that influence cavitation flow. To optimize the cavitation flow of the pumps, the flow laws near the RBSS and generation mechanisms of cavitation should be defined. This study used the shear stress transport (SST $k-\omega$) turbulence model based on the Zwart-Gerber-Belamri cavitation model combining with the experimental method to implement a comparative analysis of the unsteady flow fields near the RBSS under different operating states, in which two blades optimization schemes were proposed. Numerical simulation results demonstrate that the calculation errors are within permissible engineering error range. In the original blade scheme, the unsteady phenomena occur with axial turbulence intensity, shearing stress, and pressure fluctuation near the RBSS at blade inlet-edge. The changes of the unsteady phenomena are acute under different operating states, and the cavitation is enlarged in the unsteady regions. After optimization, the two blades structures effectively optimize the unsteady flow near the RBSS and improve the hydraulic performance and cavitation performance of the pumps. Conclusions obtained in the study have important implications to optimize the cavitation of axial-flow hydraulic machineries.

(Received in August 2017, accepted in March 2018. This paper was with the authors 3 months for 2 revisions.)

Key Words: Waterjet Pump, Cavitation, Unsteady Flow, Performance Optimization

1. INTRODUCTION

Propulsion systems are the core driving part of ships. Waterjet propulsion devices, in which the waterjet pumps are the core part, are widely applied in many propulsion devices. With the improvements in the theoretical design of pumps and the increased application of waterjet propulsions on ships in recent decades, the waterjet propulsions have gradually been developed as a highly reliable propulsion mode for ships with small and medium displacements, high navigational speed, and high performance. The working principles of waterjet propulsions are similar to that of the jet aircrafts. The propulsion pumps absorb water from the bottom of the carrier, spray out water with high velocity toward the back of the carrier through a nozzle, and propel the carrier forward with the thrust generated by the change in effluent and inflow momentum. The only difference between the aircrafts and the ships are the medium for propulsion, namely, jet for air versus waterjet for water. In practical situations, two methods are used to improve the propulsion efficiency of devices. The first method is by selecting waterjet pumps with superior hydraulic performance and cavitation performance on the basis of different factors, such as the ships model resistance and design speed, among others, whereas the second method is by reducing the hydraulic loss of the entire flow pass. In the study, the first method is regarded the main factor of propulsion efficiency. The airfoil geometric structure of the section surface influences hydraulic performance and cavitation performance [1]. Therefore, waterjet pumps with superior

performance should be designed according to fluid machineries theory based on previous researches.

The traditional design process of the waterjet pumps rotor blades are specifically implemented on blade pressure surfaces (chord line) and flow lines (mean camber chord). The airfoil of a blade section can be obtained by a thickening process following the classical airfoil profile series of the National Advisory Committee for Aeronautics (NACA) [2], which accords with wind tunnel experiment based on aerodynamic theory. When the NACA series is directly applied to the hydraulic machinery model, a flow separation can be observed and the complicated unsteady flows can be produced near the rotor blades suction surface (RBSS). In medium-speed and high-speed operations, the separated flows near the RBSS form a bound vortex that is rapidly separated; then, highly complicated turbulent flows (vortex and secondary backflows) appear until a cavitation phenomenon occurs under severe circumstances, and this degrades impeller performance and affects normal equipment operation [3, 4]. Therefore, in this study, we focus on how to accurately obtain the unsteady flow law of the RBSS under cavitation and define the internal relationship between the geometric structure of the suction surface and the unsteady flow law.

2. STATE OF THE ART

Scholars all over the world have conducted considerable explorations on the hydraulic design of waterjet pumps. The development history and achievements of waterjet propulsion devices were discussed in London in 1994. Since then, the structural form of waterjet propulsion devices has gradually transformed from screw impellers to waterjet pumps. The hydraulic design theory has also gradually developed from simple unitary design and binary design theories to the rather complicated ternary flow design types.

Wu et al. [5] obtained the distribution laws of the turbulence energy of the vane-top leakage and internal flow process of waterjet pumps with particle image velocimetry (PIV) and analysed the causes of turbulent vortex formation. Bonaiuti et al. [6] combined inverse design and computational fluid dynamics (CFD) numerical simulation with test analysis to investigate the design problems of waterjet pump parameters. Bulten [7] analysed the relationship between inlet heterogeneity and waterjet pump performance by quantifying axial velocity and the total distortion coefficient. Ji et al. [8, 9] used unsteady numerical simulation to analyse the heterogeneity effect of incoming inlet flows on the development process of internal cavitation of submersible waterjet pumps, and they verified the accuracy of the numerical simulation with a model test. Duerr and Ellenrieder [10] studied the influence of changes in revolving speed on the inlet heterogeneity of waterjet pumps. Altosole et al. [11, 12] successfully predicted the performance of waterjet pumps through numerical simulation programming and proposed two dimensionless parameters to the design process. Takai et al. [13] used CFD in the performance prediction of existing waterjet pumps and realized the optimization design of waterjet propulsion systems for high-speed ships. Cao et al. [14] used the ternary design method of velocity circulation to study the effects of blade quantity, flow direction, circular rector at central position, and circular rector at the blade outlet on impeller performance, and then designed an axial waterjet pump with good cavitation performance.

Waterjet pump rotor blades can be estimated by 3D airfoil profiling, and this technique was adopted as a research idea for airfoil profile optimization on the basis of hydrodynamic characteristics. Research scholars have conducted many research initiatives on 3D airfoil profiling. Kim et al. [15] accurately and effectively captured the cavitation phenomenon of an airfoil profile with a homogeneous mixture model based on Merkle's cavitation model and verified this method through testing. Then, the acoustic analogy Lighthill theory was used for the predictive analysis of water noises caused by cavitation. Kravtsova et al. [16] conducted a

visualization research on the flow status of cavitation using a flat plate with semi-circular leading edge and NACA10015 hydrofoil by employing high-speed photography. Goundar et al. [17] combined numerical simulation and testing to study the cavitation performance of HF-Sx hydrofoils, which were optimized to use in marine turbine propulsion devices. On the basis of a study on the cavitation characteristics of large vertical axial flow pumps, Yan et al. [1] found that changes in the geometric structure of the RBSS could alter pump cavitation performance, especially the maximum thickness reposition of airfoil profiles on the blade section, which was needed to improve pump cavitation performance.

In the abovementioned blade design processes, the airfoil profile of the RBSS required thickening in accordance with the airfoil profile principles of the NACA series. However, the effect of changing certain variables (i.e., adjusting the geometric structure of airfoil profile while retaining pressure surface(chord line) or flow lines(mean camber chord) of waterjet pumps) on cavitation performance was generally unknown, as only a few research was available on this aspect.

In this study, we selected a waterjet pump as the research object. Based on the ternary flow theory of fluid machinery, CFD technology and model experiments were combined to study the NACA series-based rotor blade and the change laws of the separated flow after adjusting the geometric structure of the airfoil profile. The change laws of turbulence intensity, Reynolds stress, and pressure fluctuation near the blade in cavitation status were analysed. Then, the cavitation formations induced by the separated vortex near the RBSS were investigated, the instability laws of fluid flow in cavitation status were determined, and optimization measures were proposed. Research conclusions could provide reference for the optimization of cavitation performance of blade-type hydraulic machineries.

The remainder of this study is arranged as follows: section 3 presents the structure of the waterjet pump and the corresponding hydraulic and test models. Section 4 emphasizes the comparative analysis of unsteady flows inside the models by CFD technology and presents two model optimization schemes. The final section summarizes this study and provides relevant conclusions.

3. METHODOLOGY

3.1 Waterjet pump model

The 3D computational domain of the waterjet pump is shown in Fig. 1. The design parameters are as follows: flow quantity of $Q_{des} = 35 \text{ m}^3/\text{h}$, revolving speed of $n = 1000 \text{ r/min}$ and head of $H = 0.40 \text{ m}$.

Figure 1: Geometric and numerical calculation models.

The main geometric parameters of the impeller and the guide vane are as follows: impeller diameter of $D_2 = 112$ mm, hub diameter of $D_h = 50$ mm, number of impeller blades at $Z_i = 4$, and number of guide vanes at $Z_v = 5$.

3.2 Computing method

To determine the turbulence characteristics of the waterjet pump, the finite volume method was used for the discretization of the 3D incompressible fluid governing equation which was performed through ANSYS CFX 16.0 computing [6]. The governing equations of the 3D turbulence numerical simulation considered the cavitation model based on the two-phase flow hybrid model, Reynolds-averaged Navier-Stokes (RANS) equation, and on the shear stress transport (SST $k-\omega$) turbulence model suited for fluid separation [18]. The discretization control of the governing equations adopted the control volume method in which the diffusion term in the equation represents the central difference scheme while the convective term represents the second-order upwind scheme. In addition, a semi-implicit pressure coupling algorithm was used to solve the equations. The Zwart equation based on Rayleigh-Plesset was used to analyse the generation and collapse of the cavitation bubbles and the mass transfer in fluid.

The computing method also considered the following design: pump inlet was the total pressure inlet. Pump outlet was the mass flow outlet. A non-slipping wall was the wall function. Energy transfers between the rotating impeller and static guide vane components were connected by “Frozen rotor”. Computational convergence standard was 10^{-5} and temperature of water medium was 25°C . The near-wall parameters were computed using the wall function method, and the near-wall tangential velocity was designated as a logarithmic relation with wall shearing stress ν .

$$u^+ = \frac{U_t}{u_\tau} = \frac{1}{\kappa} \ln(y^+) + C \quad (1)$$

where $y^+ = \rho \Delta y u_\tau / \mu$ and $u^+ = (\tau_w / \rho)^{1/2}$; u^+ is the near-wall velocity (m/s). u_τ is the friction velocity (m/s). U_t is the tangential velocity of wall surface Δy away from the wall surface (m/s). y^+ is the distance to the wall surfaces. κ is the Karman constant and C is the logarithmic layer constant related to wall surface roughness.

Table I: Grid independence verification.

Grid No.	Grid quantity/(10,000) No. of cells/thousands		$H_{\text{Des-i}}/\text{m}$
	Impeller	Vanes	
Grid 1	15.8	12.2	0.308
Grid 2	30.3	27.4	0.353
Grid 3	63.7	43.7	0.391
Grid 4	85.3	56.5	0.403
Grid 5	102.9	89.2	0.410

ANSYS ICEM 16.0 was used for the structured grid generation of the hydraulic components, in which the y^+ value was appropriately adjusted. The viscous sublayers of the near-walls of the main components that could influence hydraulic performance satisfied the $y^+ \leq 5$ requirement, whereas the other flow passage components of the near-wall viscous sublayers of the other flow passage components satisfied the $y^+ \leq 50$ requirement of the turbulence model. Grid independence verification was implemented for the computational

domain of the water pump (Table I). In Fig. 2 a, $H_{\text{Des-5}}$ represented the computed head of the pump (Grid 5). When the grid quantity of the hydraulic components was changed from small to large, the head gradually became small. The pump head increased by 1.03 % when total grid quantity was about 1,418,000, and thus, this number was applicable for the pump model (Fig. 2 b).

Figure 2: Grid dependency analysis.

3.3 Establishment of test model

Considering that the study object had a small head, the generated head that operated in large flow quantities could not overcome pipeline resistance. Consequently, the complete performance curves could not be obtained. To solve this problem, an auxiliary pump was added to the pump testing apparatus. To further improve measurement accuracy, two pressure measurement apparatuses with double-layer pipe wall structures were added to the inlet and outlet of the axial flow-type micro-head waterjet pump. Four uniformly distributed holes were set on the internal section to link the lateral pressure measurement holes. The hydraulic performance and cavitation performance of the pump were determined by regulating the opening of the inlet and outlet valves of the pump. The general design of the testing apparatus and pictures from field test are shown in Fig. 3 a.

Figure 3: Testing apparatus schematic and pictures of the waterjet pump.

To verify the accuracy of numerical simulation, the testing apparatus was built based on CNS (GB 3216-2005) following grade 1 precision of international standards (ISO 9906-1999) [19]. The measuring equipment achieved high measurement accuracies. The pressures on the pump inlet and pump outlet were measured with a pressure sensor (Rosemount 3051), and measurement accuracy was $\pm 0.075\%$. Flow quantity was measured with an electromagnetic flowmeter (LDG-80), and the accuracy was $\pm 0.1\%$. Motor shaft power was measured with a revolving torque sensor (CYT-302), and the accuracy was $\pm 0.2\%$. An uncertainty analysis of the measurements of the entire testing apparatus was conducted, and the results showed that the uncertainty was within the permissible range of international standards.

4. RESULT ANALYSIS

4.1 Performance comparison

Fig. 4 a shows the comparative results of pump hydraulic performance from the numerical simulation and testing. The calculation result presents a similar change trend with the numerical simulation. The computed pump head is 0.415 m for the design condition, and the error compared with test head (0.40 m) is 3.75 %. Computational efficiency is 58.03 %, and the error compared with test efficiency (55.32 %) is 4.90 %. The computational accuracies of the other computational conditions also satisfy engineering application requirements.

Fig. 4 b shows the comparative results of pump cavitation performance from the numerical simulation and testing. A cavitation occurs in the pump unit when the pump inlet pressure is changed, i.e., when the head decreases by 3 % in relation to the head of the design operating point (critical cavitation point) [20]. Under the design condition and different net positive suction head available ($NPSH_a$), the result computed by CFD is slightly larger than the test result, but the change trends are almost identical. From $NPSH_a \geq 5$ m, the head remains somewhat unchanged when the $NPSH_a$ decreases. When $NPSH_a < 5$ m, the head rapidly decreases. When $NPSH_a = 4.71$ m, the head decreases by 3.02 %, a close approximation of the critical cavitation point. Meanwhile, the error for the net positive suction head required ($NPSH_r$) of the pump ($NPSH_r = 4.55$ m) obtained through the test is 3.64 %. Therefore, CFD-computed results of the hydraulic performance and cavitation performance accurately predict the real flow status inside the pump.

a) Hydraulic performance curves of model pump b) Cavitation performance curves of model pump
Figure 4: Comparison of performance between numerical simulation and field testing.

4.2 Analysis of turbulence intensity at the impeller surface

Turbulence intensity is an important evaluation indicator of unsteady motion characteristics inside fluids. In this study, I_w is used to determine the turbulence intensity inside the pump, as

shown in Eq. (2), and to which the defined $r_b = x/X$ is illustrated in Fig. 6. The x -coordinate, which corresponds to the ratio of sectional position to chord length of the airfoil profile on the blade section, is the radial position; w'_{rms} , the variance root of axial transient fluctuation velocity, is the turbulence intensity of the fluid on axial flow direction; and $\bar{W}$ is the time-average axial velocity of the fluid flow.

$$I_w = \frac{w'_{rms}}{\bar{W}} \quad (2)$$

Figure 5: Change inside the impeller under different conditions.

Figure 6: Position of inside the impeller.

Fig. 5 shows the distribution of axial fluid intensity I_w inside the impeller. When $\delta_Q = 1.0$ and $NPSH_a = 9.98$ m, some segment of the unsteady regions for I_w value appear near the impeller inlet. When the fluid flows from the impeller inlet to the outlet, the I_w value of the pressure surface fluctuates but only within a small range, and the changes in I_w remarkably increase near the RBSS. This finding indicates a noticeable instability near the RBSS. The I_w value gradually decreases when the fluid flows from the impeller hub to the impeller flange, which suggests that fluid turbulence is strong at the impeller hub.

Cavitation occurs inside the pump when $NPSH_a = 4.55$ m, although the change in I_w value is negligible near the pressure surface. However, the I_w value sharply increases at RBSS by nearly 10 times compared with that of the design condition, and the increase mainly manifests near the impeller hub inlet and outlet, especially along the surface close to the blade outlet. These positions are mainly influenced by different factors, such as collapse of cavitation bubbles in the fluid near the RBSS and the dynamic and static coupling of impeller and guide vanes. The turbulence intensities on those positions are strong.

4.3 Reynolds stress analysis

A viscous fluid bears Reynolds stress aside from pressure, gravity, and intermolecular force. Reynolds stress, which is generated as a convective term of the N - S equation, is expressed as $-\rho \overline{u'_i v'_j}$. Furthermore, Reynolds stress represents the influence of turbulent fluctuation on time-average flow. The term $\overline{u'_i v'_j}$ denotes the correlation degree among different fluctuating velocity components at the same spatial point at single moment, and it is a second-order symmetric tensor in a rectangular coordinate system, which can be expressed as:

$$-\overline{u'_i v'_j} = \begin{vmatrix} \overline{u'u'} & \overline{u'v'} & \overline{u'w'} \\ \overline{v'u'} & \overline{v'v'} & \overline{v'w'} \\ \overline{w'u'} & \overline{w'v'} & \overline{w'w'} \end{vmatrix} \quad (3)$$

In the above equation, the tangential stress term is a term in which its component on the diagonal line is excluded. Given that the flow of fluid inside the pump in the x-directions and y-directions (tangential direction) is symmetrical, the $-\overline{u'v'}$ tangential stress term is considered for the convenience of the study. Similar change laws are employed for the other directions. The dimensionless shearing stress $-\overline{u'v'}/\overline{W^2}$ is defined to reflect the turbulence characteristics of the viscous fluid inside the pump.

Figure 7: Change of shearing stress coefficient inside the impeller under different conditions.

Fig. 7 illustrates the analytical results of the shearing stress coefficient inside the impeller under different conditions, in which the x-coordinate is $-\overline{u'v'}/\overline{W^2}$. When $\delta_Q = 1.0$ and $NPSH_a = 9.98$ m, from the impeller inlet to outlet, the $-\overline{u'v'}/\overline{W^2}$ value inside the impeller gradually increases and fluctuates between -0.4 and 0.4 . When $\delta_Q = 1.0$ and $NPSH_a = 4.55$ m, the $-\overline{u'v'}/\overline{W^2}$ value changes significantly near the inlet and fluctuates in the 0 – 1.0 range, a finding that is similar to the case of cavitation, i.e., cavitation increases the shearing stress inside the pump.

4.4 C_p distribution

The internal velocity and pressure fields of the waterjet pump are used as the main parameters of flow analysis. The energy conversion inside the pump is a mutual conversion between kinetic energy influenced by velocity field and pressure energy, which in turn is influenced by pressure field. The internal pressure of the pump directly affects the pump head, and the

pressure fluctuation coefficient directly influences the safety and stability of pump operation. Fig. 8 shows the distribution of pressure coefficient C_p at the impeller surface with different $NPSH_a$ values under the design working condition. In Eq. (4), the y-coordinate represents the change in axial distance from the impeller inlet to the outlet given the same radius.

$$C_p = 2(p - \bar{p}) / \rho \bar{W}^2 \quad (4)$$

where p is the transient pressure value on the blade surface, while $\bar{p}$ is the average pressure on the blade section. In Fig. 8, $NPSH_a = 9.98$ m. C_p distribution is regular when no cavitation occurred in the pump. From the pressure inlet to the pressure outlet, C_p decreases and the fluctuation is significant near the impeller inlet, especially when $C_p = 0.9$. From the impeller hub to the impeller flange, C_p gradually increases, but the range is small. Near the suction surface, C_p is stable and pressure fluctuation is large. With $NPSH_a$ decreased, the range of pressure fluctuation coefficient near the RBSS expands after the pump reaches the critical cavitation pressure. Unlike in the case where in $NPSH_a = 9.98$ m, a certain effect at the suction surface of the blade outlet is observed. A vortex appears at the impeller outlet mainly on the influence of fluid separation near the outer edge of the impeller hub, and the internal pressure fluctuation of the impeller changes significantly due to the dynamic and static coupling of the impeller blades.

Figure 8: Pressure distribution coefficient C_p on the impeller surface when $\delta_Q = 1.0$.

4.5 Model optimization

According to above analysis, the unsteady regions inside the pump mainly occur near the suction surface of the blade inlet. A sharp change in turbulence intensity is observed, and the unsteady regions expand when cavitation occurs inside the pump. The optimization design of the skeleton line for RBSS is implemented, and two schemes are designed and compared with the original scheme (Fig. 9). Scheme I is the original impeller. Scheme II represents the design in which the flow line on the blade suction surface of the airfoil profile is changed, but the flow line on blade pressure surface is unchanged, and $\phi = 0.7$, where ϕ is the ratio of distance from the maximum thickness $\delta_{\max}$ position of the blade to the inlet edge on the chord length of airfoil profile, while the other conditions are retained. Scheme III represents the blade shape in which the flow lines along the suction surface and pressure surface are paralleled, but the flow line on the blade pressure surface is unchanged, i.e., the relative velocity near the RBSS is equal to that of the working surface (w_2 are equal in two surfaces) as shown in Fig. 9.

Figure 9: Optimization schemes.

Fig. 10 shows the hydraulic performance curves of a pump for the three different impeller schemes. The hydraulic performances of schemes II and III are superior to that of scheme I. Under the design working condition, the head of scheme II is $H = 0.44$ m, which is 7.32 % higher than that of scheme I. The head of scheme III is $H = 0.48$ m, which is 17.07 % and 8.33 % higher than those of schemes I and II, respectively. Moreover, the efficiency of scheme II is $\eta = 61.20$ %, which is 3.17 % higher than that of scheme I. The efficiency of scheme III is $\eta = 61.78$ %, which is 3.75 % and 0.58 % higher than those of schemes I and II.

Figure 10: Hydraulic performance curves.

Figure 11: Cavitation performance curves.

To comprehensively investigate the hydraulic performance of the large-scale axial flow pump under the different schemes, the cavitation performance is assessed initially. The three cavitation performance curves are shown in Fig. 11. The computations of the three schemes are all implemented under the design working conditions ($\delta_Q = 1.0$). The head decreases by $\delta_Q = 3.12$ % in scheme II when $NPSH_a = 4.21$ m, a working condition that approximates the critical cavitation, and its $NPSH_a$ decreases by 0.36 m in relation to scheme I ($NPSH_a = 4.57$ m). The head decreases by $\delta_Q = 3.04$ % in scheme III when $NPSH_a = 5.13$ m, a working condition that approximates the critical cavitation, and its $NPSH_a$ increases by 0.56 m in relation to scheme I.

Fig. 12 shows the distribution of pressure fluctuation coefficients C_p near the RBSS for the different schemes. In relation to scheme I, the C_p distribution at the pressure surface of scheme II slightly changes. The value of the C_p distribution near the RBSS of scheme II is significantly larger than that of scheme I, although significant fluctuations appear in these two regions. In relation to schemes I and II, the C_p values of scheme III slightly decrease at the blade inlet to improve the steadiness of the fluid flow on this position, but change significantly near the suction surface. In relation to scheme I, the C_p value of scheme III only changes slightly at the centre of the RBSS (approximately one constant value), while the value at the blade outlet is larger for the same position in scheme III compared with that of scheme I. Furthermore, in relation to scheme II, the C_p values near the suction surface and range of unsteady region of scheme III are both optimized.

Figure 12: Comparison of distribution of pressure fluctuation coefficients C_p .

Scheme II effectively improves the hydraulic performance and cavitation performance of the pump, but the improvement of cavitation is only minimal. Scheme III improves the hydraulic performance of the pump very well, but the $NPSH_a$ value increases, which suggests that cavitation performance is poor. Thus, the scheme III method can be used to design large-scale axial flow pump blades with high requirements for hydraulic performance but low requirements for cavitation performance. By contrast, the design method of scheme II can be used to optimize large-scale blades with high requirements for cavitation performance.

4. CONCLUSION

To improve the cavitation performance and determine the relationship between the geometric structure of the RBSS and unsteady flows of the waterjet pump under different working conditions, the variation in turbulence intensity, Reynolds stress, and pressure fluctuation near the RBSS were discussed by numerical simulation. Moreover, two methods were proposed to optimize the cavitation performance. The following conclusions are drawn.

(1) The comparative results from numerical simulation and experiment show that errors of head, efficiency, and critical $NPSH_a$ of the pump under the design working condition are 3.75 %, 4.90 %, and 3.64 %, respectively. Computational accuracy of numerical simulation meets engineering application requirements.

(2) The turbulence intensity, Reynolds stress, and pressure fluctuation on the blade surface of the waterjet pumps have been analysed. The flow law near the RBSS is closely related to

the airfoil profile of the blade, especially at the maximum thickness position of the airfoil profile. Unsteady flows occur near the inlet of the RBSS. Appearing cavitation will enlarge the unsteady flow range region.

(3) Adjusting the airfoil profile structure of the rotor blades, which are proposed in the study, can effectively improve hydraulic performance of the waterjet pumps. Moreover, repositioning the maximum thickness of the airfoil profile on the rotor blade can effectively improve the hydraulic performance and cavitation performance.

This study combines numerical simulation and experiments, establishes the internal relationship between the geometric structures of the RBSS and unsteady flows, and proposes guidelines for the optimization of waterjet pumps performances. However, the conclusions of this study still have limitations on optimizing the detailed structures of the RBSS owing to the lack of complete measured data after model optimization. Further experimental studies should be carried out by establishing the multi-objective optimization of airfoil profile near the RBSS to verify the correctness of these methods.

ACKNOWLEDGEMENT

The study was supported by Fundamental Research Funds for the Central universities (Grant no. JZ2015HGBZ0469 and JZ2015HGBZ0128).

REFERENCES

- [1] Yan, H.; Liu, M.-Q.; Liang, X.; Lin, P. (2014). Numerical simulation on cavitation characteristics of large-scale axial-flow pumps based on orthogonal experiment, *Journal of Huazhong University of Science and Technology*, Vol. 42, No. 12, 695-701, doi:[10.13245/j.hust.141208](https://doi.org/10.13245/j.hust.141208)
- [2] Shi, W.-D.; Zhang, J.-J.; Zhang, D.-S.; Zhao, R.; Zhang, L. (2016). Numerical simulation of accelerated flow on hydrofoil cavitation, *Transactions of the Chinese Society for Agricultural Machinery*, 2016, Vol. 47, No. 4, 1-7, doi:[10.6041/j.issn.1000-1298.2016.04.001](https://doi.org/10.6041/j.issn.1000-1298.2016.04.001)
- [3] Ji, B.; Lou, X.-W.; Wu, Y.-L.; Peng, X.-X.; Duan, Y.-L. (2013). Numerical analysis of unsteady cavitating turbulent flow and shedding horse-shoe vortex structure around a twisted hydrofoil, *International Journal of Multi-phase Flow*, Vol. 51, 33-43, doi:[10.1016/j.ijmultiphaseflow.2012.11.008](https://doi.org/10.1016/j.ijmultiphaseflow.2012.11.008)
- [4] Peng, X. X.; Ji, B.; Cao, Y.-T.; Xu, L.-H.; Zhang, G.-P.; Luo, X.-W.; Long, X.-P. (2016). Combined experimental observation and numerical simulation of the cloud cavitation with U-type flow structures on hydrofoils, *International Journal of Multiphase Flow*, Vol. 79, 10-22, doi:[10.1016/j.ijmultiphaseflow.2015.10.006](https://doi.org/10.1016/j.ijmultiphaseflow.2015.10.006)
- [5] Wu, H.; Miorini, R. L.; Katz, J. (2011). Measurements of the tip leakage vortex structures and turbulence in the meridional plane of an axial water-jet pump, *Experiments in Fluids*, Vol. 50, No. 4, 989-1003, doi:[10.1007/s00348-010-0975-0](https://doi.org/10.1007/s00348-010-0975-0)
- [6] Bonaiuti, D.; Zangeneh, M.; Aartojarvi, R.; Eriksson, J. (2010). Parametric design of a waterjet pump by means of inverse design, CFD calculations and experimental analyses, *Journal of Fluids Engineering*, Vol. 132, No. 3, Paper 031104, 15 pages, doi:[10.1115/1.4001005](https://doi.org/10.1115/1.4001005)
- [7] Bulten, N. W. H. (2006). *Numerical analysis of a waterjet propulsion system*, PhD Thesis, Department of Mechanical Engineering, Technische Universiteit Eindhoven, doi:[10.6100/IR614907](https://doi.org/10.6100/IR614907)
- [8] Ji, B.; Luo, X.-W.; Wang, X.; Peng, X.-X.; Wu, Y.-L.; Xu, H.-Y. (2011). Unsteady numerical simulation of cavitating turbulent flow around a highly skewed model marine propeller, *Journal of Fluids Engineering*, Vol. 133, No. 1, Paper 011102, 8 pages, doi:[10.1115/1.4003355](https://doi.org/10.1115/1.4003355)
- [9] Ji, B.; Luo, X.-W.; Peng, X.-X.; Wu, Y.-L.; Xu, H.-Y. (2012). Numerical analysis of cavitation evolution and excited pressure fluctuation around a propeller in non-uniform wake, *International Journal of Multiphase Flow*, Vol. 43, 13-21, doi:[10.1016/j.ijmultiphaseflow.2012.02.006](https://doi.org/10.1016/j.ijmultiphaseflow.2012.02.006)
- [10] Duerr, P.; von Ellenrieder, K. (2013). Investigation of non-uniform waterjet pump inflow for a range of ship speeds, *Oceans 2013 MTS/IEEE*, 7 pages

- [11] Altosole, M.; Benvenuto, G.; Figari, M.; Campora, U. (2004). Performance prediction of a waterjet propelled craft by dynamic numerical simulation, *Proceedings of the 4th International Conference on Waterjet Propulsion*, London
- [12] Altosole, M.; Benvenuto, G.; Figari, M.; Campora, U. (2012). Dimensionless numerical approaches for the performance prediction of marine waterjet propulsion units, *International Journal of Rotating Machinery*, Vol. 2012, Paper 321306, 12 pages, doi:[10.1155/2012/321306](https://doi.org/10.1155/2012/321306)
- [13] Takai, T.; Kandasamy, M.; Stern, F. (2011). Verification and validation study of URANS simulations for an axial waterjet propelled large high-speed ship, *Journal of Marine Science & Technology*, Vol. 16, No. 4, 434-447, doi:[10.1007/s00773-011-0138-x](https://doi.org/10.1007/s00773-011-0138-x)
- [14] Cao, Y.-L.; Wang, Y.-S.; Jin, S.-B. (2015). Three-dimensional hydraulic design of axial waterjet pump, *Journal of Traffic and Transportation Engineering*, Vol. 15, No. 2, 42-49
- [15] Kim, S.; Cheong, C.; Park, W.-G.; Seol, H. (2016). Numerical investigation of cavitation flow around hydrofoil and its flow noise, *Transactions of the Korean Society for Noise and Vibration Engineering*, Vol. 26, No. 2, 141-147, doi:[10.5050/KSNVE.2016.26.2.141](https://doi.org/10.5050/KSNVE.2016.26.2.141)
- [16] Kravtsova, A. Y.; Markovich, D. M.; Pervunin, K. S.; Timoshevskiy, M. V.; Hanjalić, K. (2014). High-speed visualization and PIV measurements of cavitating flows around a semi-circular leading-edge flat plate and NACA0015 hydrofoil, *International Journal of Multiphase Flow*, Vol. 60, 119-134, doi:[10.1016/j.ijmultiphaseflow.2013.12.004](https://doi.org/10.1016/j.ijmultiphaseflow.2013.12.004)
- [17] Goundar, J. N.; Ahmed, M. R.; Lee, Y.-H. (2012). Numerical and experimental studies on hydrofoils for marine current turbines, *Renewable Energy*, Vol. 42, 173-179, doi:[10.1016/j.renene.2011.07.048](https://doi.org/10.1016/j.renene.2011.07.048)
- [18] Shi, W.-D.; Zhang, D.-S.; Guan, X.-F.; Leng, H.-F. (2010). Numerical and experimental investigation of high-efficiency axial-flow pump, *Chinese Journal of Mechanical Engineering*, Vol. 23, No.1, 38-44, doi:[10.3901/CJME.2010.01.038](https://doi.org/10.3901/CJME.2010.01.038)
- [19] ISO 9906 (2012). *Rotodynamic Pumps – Hydraulic performance acceptance tests – Grades 1, 2 and 3*, 2nd edition, Geneva, 59 pages (<https://www.iso.org/standard/41202.html>)
- [20] ANSI/API STD 610 (ISO 13709:2009 Identical Adoption) (2010). *Centrifugal pumps for petroleum, petrochemical and natural gas industries*, 11th edition, American Petroleum Institute, Washington, DC