Numerical study on hydraulic and self-priming performance of a double-stage self-priming pump

H Y Yao¹, Y D Zhang¹, D Z Wu¹,²,* and P Wu¹

¹ College of Energy Engineering, Zhejiang University, Hangzhou, 310027, China
² The State Key Laboratory of Fluid Power Transmission and Control, Zhejiang University, Hangzhou, 310027, China

Email: wudazhuan@zju.edu.cn

Abstract. Vertical self-priming pump has numerous advantages and has been widely used in industrial and agricultural areas. Nevertheless, impeller in single-stage self-priming pump bears large diameter and huge friction loss in the delivery of high-lift fluid, and enhanced requirement for sealing capacity meanwhile generates greater increment of power loss from sealing auxiliary-impeller and sealing ring. Both of these have resulted in low efficiency in operation, to the extent of imposing restriction on the further application of self-priming pump. In this paper, a new design of double-stage vertical self-priming pump with two impellers settled back to back was put forward to cover the shortfalls above. The internal flow field was calculated adopting realizable k-ε turbulence model in commercial CFD software Fluent and performance prediction was conducted. Concurrently, two-phase flow inside this self-priming pump without backflow hole was calculated by gas-liquid two-phase unsteady simulation to explore the self-priming characteristics. In order to consist with practical self-priming situation, a section of suction pipe filled with air was installed as the initial condition, pressure inlet and pressure outlet was set as boundary condition. Base on the results, the variation of pressure distribution and the fluctuation of air volume fraction on the specific cross-section of impeller and volute were obtained to study the self-priming process. The simulation and test results showed distinct uniformity, which indicated that the optimizing design provided impacts great effect on efficiency improvement as well as meeting the purpose of energy-saving and expense-reduce in the aspect of self-priming performance. The research here has provided the instruction to the design of high-head vertical self-priming pump.

1. Introduction

The traditional vertical self-priming pump, belonging to external-mix self-priming pump, was developed on the basis of horizontal self-priming pump. Except for the first use, it has no need of priming pump and the special structure of the pump can help exhaust out the air in suction pipe at start-up stage. The self-priming process can be described as below. After the motor is started, impeller rotates in high speed and pressurizes the liquid stored in it out to volute. Meanwhile, vacuum is generated at the entrance of the impeller, which drives the air from suction pipe to mix with water in the volute. Then the air-water mixture flood into separation chamber and slow down. Under the effect of baffle, the two phases with low velocity separate with each other. Air, of low density relatively, emerge above and discharge from outlet pipe at upper location, and water, of higher density, sink down to blend with the air in volute through backflow hole. Repeating this step for several minutes,
the air in the suction pipe is exhausted and the machine turn into the common water transmission condition.

Recent year, many researchers have carried out plenty of study on the self-priming pump experimentally and numerically. Nevertheless, the majority of them focused on the energy conservation reconstruction and two-phase simulation in one-stage self-priming pump, with little effort at the researches concerning multistage self-priming pump. Gong E et al. [1] found the problem of siphonage and excessive self-priming time in traditional vertical model. Confronted with these defects, the structure of a new type of non-leakage vertical self-priming pump was improved by adopting special entrance form and fitting impeller and vane together. Nevertheless, these transformations inevitably cause a increment on hydraulic losses. Sun Y et al. [2] optimized hydraulic components and matching structure on the basis of traditional WFB pump to reduce hydraulic losses and used multistage auxiliary-impeller to reduce mechanical losses while improving sealing performance. Exploration on the mechanism of self-priming process related to two-phase flow by unsteady numerical simulation method also became active. In Huang S et al.’s paper [3], the time-related gas–liquid two-phase distribution, the pressure and velocity were computed and analyzed, which could be used to estimate the self-priming time. For better evaluating the accurate degree of numerical results, experimental researches (Lu T et al. [4] recorded internal flow field using high-speed photography, Wang T et al. [5] established transient test system to study transient effect in self-priming process) have gotten many achievements. All above have provided the foundation for improvement of numerical computation technique in self-priming performance prediction and optimization design.

In high-head working condition, numerous problems have exposed using ordinary self-priming pump, for instance, unreasonable matching structures between volute and separation chamber, excessive power consumption for large-sized auxiliary impeller [6], tremendous volume losses as the increase of leakage flow rate through sealing clearance, poor hydraulic efficiency of impeller in low specific-speed et al. All of the above resulted in poor performance, and excess of power consuming than design specification. In this paper, a brand-new design of double-stage vertical self-priming pump was introduced to meet the requirement in high-head working condition. The hydraulic performance and self-priming process were investigated with CFD method.

2. Structure of vertical self-priming pump and performance parameters

The pump (figure 1) consists of several primary components as follows, impeller (two stages), volute(two stages), auxiliary impeller, separation chamber, shaft, magnetic valve, suction pipe, exhaust pipe, motor et al. In normal water transmission condition, the fluid passed the goose-neck-shaped suction pipe and enters the first-stage impeller. After being pressurized in impeller and diffusing in volute, the high-pressure liquid came into separation chamber in lower layer. Then it got into separation chamber in upper layer through connecting hole, and was inhaled by second–stage impeller settled reversely. The liquid, that was pressurized twice, would be exhausted out of the pump directly through second-stage volute and outlet pipe, two of which are interconnected. This design can reduce the sealing power consumption of auxiliary impeller and cut down the leakage from the inlet clearance.

The geometric parameters and hydraulic specification required are shown in table 1 below.

| Geometric Parameters | Hydraulic Specification |
|----------------------|-------------------------|
| Suction diameter $D_1$ | First stage 179(mm)  Second stage 192(mm)  Flow rate $Q_0$ 370(m³/h) |
| Impeller outer diameter $D_2$ | Total head $H$ 47(m) |
| Export diameter $b_2$ | Power $P$ 75(kW) |
| Blade outlet angle $\beta_2$ | Rotating speed $n$ 1480(rpm) |
|                      | Suction time $T$ 180(s) |
There is optimal specific speed for turbomachinery, in which the impeller has maximum hydraulic efficiency at design point [7]. For centrifugal pump, the optimal specific speed is in the range of 120 to 180. While the specific speed of one-stage impeller for the hydraulic parameters above is 96.4. In another respect, according to the special structure proposed in this paper, the sealing pressure for auxiliary impeller is the pressure of fluid exhausted from one stage. It is beneficial to reduce sealing power consumption when decrease the discharge head by first-stage impeller properly. Ultimately, the design head for the impellers is 20m and 35m respectively, with specific speed of 183.1 and 120.4.

3. Steady numerical simulation method

3.1. Domain and mesh

The 3D model of computational region including sealing clearance and flow channel around auxiliary impeller (figure 2) was established using UG NX. On the account of structural complexity, the whole 3D computational domain was separated into several parts and each part was imported into ICEM to finish mesh division respectively. The unstructured mesh was generated for domain in complicated structure and big size like impeller, volute, auxiliary impeller and separation chamber (figure 3(b)). Structure mesh with smaller size was taken in the mesh generation of sealing clearance (figure 3(c)) and suction pipe to achieve the balance between mesh quantity and computing accuracy.

To ensure the accuracy of numerical simulation, a grid independence test has been carried out and simulations with grid number of 3128649, 3860518 and 4663289 were solved and compared. The predicted total head for three grid sizes are presented in table 2 which shows that the difference of predicted head were within 1.5% and the influence of grid was negligible. Base on these analyses, all subsequent simulations in this paper were carried out with the mesh quantity of 3860518 and the final element of the solution domain are shown in figure 2(a).

Table 2. Total head caculated by different grid numbers.

| Grid | Grid number | Total head(m) |
|------|-------------|---------------|
| 1    | 3128649     | 48.68         |
| 2    | 3860518     | 49.13         |
| 3    | 4663289     | 49.37         |
3.2. Algorithm and boundary condition
In this study the ANSYS Fluent commercial code was used to solve the continuity and RANS equations. Realizable k-ε turbulence model was adopted to close numerical equations, and coupling numeration of velocity and stress field was based on SIMPLEC algorithm. Governing equations were discreted by finite volume method and second order upwind scheme. The rotation flow was simulated using multiple referenced coordinate. For details, the domain of impeller and auxiliary impeller were simulated by relative coordinate rotating in design speed, whereas the other static components were simulated by static coordinate. Average fluid flowing velocity was set for the inlet of computation domain according to the different working conditions and pressure boundary condition was inducted for outlet.

4. Steady simulation results and analysis
First, complete characteristics of the pump for various flow rates become available by changing inlet velocity. Test data obtained from pump performance experiment was used to verify the accuracy and applicability of simulation method mentioned above. Beyond that, in the simulation, the shroud, hub, surfaces on sealing clearance and auxiliary impeller was set as moving wall. Therefore, volume losses (calculating by the leakage from clearance) and part of disc friction losses (shear force on moving surface) can be made into consideration in this case when simulating the internal flow. From figure 3 below, we can estimate that the simulation result shows good agreement with experiment data. From figure 3 it can be found that the head gained from CFD method is higher than that from experiment. And so is the efficiency by CFD in most part of the curve. While the power consumption calculated from CFD results is similar to test data in design point and large capacity area. And the design in design point met the requirement of working demands, with margin of 5 kW to 75 kW.

Figure 2. Mesh of the double-stage self-priming pump.

Figure 3. Performance prediction and experimental results of the pump.
For the analysis of the internal flow in pump chamber, figure 4 shows that fluid pressurized by impeller slows down adequately in first-stage volute. As there is less flows impact and recirculation in separation chamber, head loss decrease to 2.6 m. In the aspect of improving hydraulic efficiency of impeller, it is important to get uniform fluid field with less flow separation and secondary flow. So that hydraulic design was based on the judgment criterion of local euler head distribution [8]. Figure 5 and figure 4 present velocity and static pressure distribution. Hydraulic efficiency for two stages comes up to 89% and 88% respectively. Figure 7 demonstrates the flow field information around auxiliary impeller in which pressure was layered distributed in the radial direction. The head generated by auxiliary impeller was higher than that by first impeller, which verified the sealing property of the auxiliary impeller.

**Figure 4.** Internal flow field on vertical central section.

**Figure 5.** Internal flow field on middle section of first-stage impeller.

**Figure 6.** Internal flow field on middle section of second-stage impeller.
5. Unsteady simulation and results analysis

The volute without backflow hole enables a high hydraulic efficiency in the way of reducing recirculation loss [9]. Nevertheless, this structure goes against self-priming performance. It is necessary to use CFD for further study on the unsteady gas-liquid two-phase self-priming process. In this paper, water and air at 20°C were taken as the liquid phase and gas phase respectively. Boundary condition for unsteady simulation was pressure inlet (Gauge pressure is zero) and pressure outlet. To better agree with the reality [10-11], the simulation procedure was initialized as a part section of the suction pipe filled with air, as shown in figure 8. To control the flow rate, throttle valve was added downstream from the outlet of pump chamber. The eulerian two-phase model and Realizable k-ε model were selected in governing equations. The unsteady Phase Couple SIMPLE implicit algorithm was selected to discrete equations. Second-order upwind scheme were set for discrete form of momentum, turbulence kinetic energy and specific dissipation rate, while first order upwind for volume fraction. The time step was made 5.6×10^-4 s, the same as rotating interval in one degree for impeller.

![Figure 7. Internal flow field on middle section of auxiliary impeller.](image)

Figure 7. Internal flow field on middle section of auxiliary impeller.

Figure 9 shows gas-liquid two-phase distribution contours on vertical central section at several typical moment during self-priming process, in which gas (void fraction equal to 1.0) is represented by red, while liquid (void fraction equal to 0) is signed by blue. Figure 10(a) is the initial state when suction pipe was filled with 3-meter-long air. At 1.51 s, shown in figure 10(b), the first-stage impeller is full of air and the air in suction pipe is exhausted into pump chamber gradually. At this moment, the first-stage impeller produces power only on gas. At 1.96 s in figure 10(c), the gas, under the drive of density difference, enter into the upper stage of separation chamber through the transitional hole. As the air blend with the water at the upper stage of pump chamber, the void fraction level in first-stage
The impeller and volute of first stage are the core components in self-priming process. In this paper, the air volume fraction contour and vector distribution on middle section at several typical moments was also presented. Figure 9(a) shows that in the early stage, the air volume fraction near suction side is higher than that near pressure side. This distribution is mainly due to the density difference. As the air has been inhaled in the revolving impeller, it lags behind the water. Therefore, the air gets close to suction side, where it is more liable to generate vortex. And water occupied the district with high-energy jet flow near pressure side. Furthermore, since the impact of centrifugal force, the heavier water was thrown off to volute wall, while the lighter air gathered around periphery of impeller. The perspective on middle section also shows that air was full of the whole impeller at 1.51 s and the impeller performed the least power at this time, which was in accordance with result from paragraph above. For the analysis of flow field, we have selected the velocity vector distribution of liquid at several times in figure 10. The vector in domain of impeller symbolize the relative velocity in rotating coordinate, and that in domain of volute symbolize the velocity in stationery reference frame. With the increase of void fraction, figure 10(a) and figure 10(b) show more vortexes appear at the tail of suction side. At 1.51 s when it comes to the highest void fraction in impeller, the flow field exhibited strong asymmetry, in which the relative velocity in flow channel close to cut-water was extremely high. From the variation among figure 10(b), (c) and (d), intensity of vortex decline as the void fraction decrease and when the void fraction was inferior to 0.4, the vortex fade away.
Figure 10. Volume fraction contour on middle section of first-stage impeller.

(a) t=0.78 s  (b) t=1.51 s  (c) t=2.18 s (d) t=3.07 s

Figure 11. Velocity vector (liquid) on middle section of first-stage impeller.

(a) t=0.78 s  (b) t=1.51 s  (c) t=2.18 s (d) t=3.07 s

Figure 12. Time history of gas volume fraction and static pressure.

(a) First stage impeller  (b) Second stage impeller
6. Conclusion
In this paper, a novel of new type double-stage self-priming pump was proposed. And the performance prediction and flow analysis on this self-priming pump were carried out by using CFD method. According to the results, we can draw the following conclusion.

Two-stage form in high-head self-priming pump can optimize the design parameters of each impeller through rational redistribution of head despite of the complexity in structure. The designed impellers and matching structures between hydraulic components in this paper achieved a good result in the application which have met the requirement of working demands, with margin of 5 kW. The correspondence between simulation results and experimental data confirmed that optimization with CFD method in complex structural self-priming pump was practicable. Under the design flow rate of 370 m³/h, the result from CFD method and experiment obtained the minimum error, which was 8% in head and 5% in efficiency.

Unsteady two-phase simulation can effectively reveal the mixing and separating process of air and water after start-up. The relationship between void fraction and generation of vortex in the impeller inner flow field has been presented. The air void fraction can reach 9.7% and 4.7% respectively in two impeller at some moment of the early stage. In first impeller, the increase of air void fraction and degradation on performance ability change consistently. Whereas in second impeller, the performance ability and air void fraction was influenced by the condition of first impeller which showed a more complicated fluctuation. The result and analysis in this paper can provide basic information for further investigation.

Acknowledgments
This work was supported by the National Natural Science Foundation of China (Grant No. 51606165) and the Fundamental Research Funds for the Central Universities. The supports are gratefully acknowledged.

References
[1] Gong E X, Tian H P, Wang R S, Li Q and Song W W 2008 Structural characteristics and design improvements of non-leakage vertical self-priming pump Journal of Drainage and Irrigation Machinery volume 26 issue 2 pp 36-39
[2] Sun Y B, Chen T, Yang S, Wu D Z and Wang L Q 2013 Improvement design of hydraulic components and structure of vertical self-priming pump Journal of Zhejiang University (Engineering Science) volume 47 issue 2 pp 332-338
[3] Huang S, Su X, Guo J and Yue L 2014 Unsteady numerical simulation for gas-liquid two-phase flow in self-priming process of centrifugal pump Energy Conversion and Management volume 85 issue 9 pp 694-700
[4] Lu T Q, Li H, and Zhan L 2016 Transient numerical simulation and visualization of self-priming process in self-priming centrifugal pump Journal of Drainage and Irrigation Machinery volume 34 issue 11 pp 927-933
[5] Wang T and Xu Z Q 2013 Experimental Research on the Transient Hydraulic Performance of Irrigation Pumps during the Self-priming Period China Rural Water and Hydropower volume 5 pp 57-60
[6] Wu D Z, Yang S, Sun Y B, Wang L Q and Min W B 2012 Multistage auxiliary-impeller performance analysis and its application Journal of Drainage and Irrigation Machinery volume 30 issue 1 pp 15-19
[7] Brennen C E 1994 Hydrodynamics of Pumps (UK: Concepts Eti & Oxford University Press) p 15
[8] Yan P, Chu N, Wu D Z, Cao L L, Yang S and Wu P 2017 Computational fluid dynamics-based pump redesign to improvement efficiency and decrease unsteady radial forces Journal of fluids Engineering volume 139 pp 011101-1-011101-11
[9] Xia L, Wu P, and Wu D Z 2015 Effects of reflux hole on the performance of self-priming pump. *Chinese Journal of Engineering Design* volume 22 issue 3 pp 284-287

[10] Noghrehkar G R, Kawaji M, Chan A M C, Nakamura H and Kukita Y 1994 Investigation of centrifugal pump performance under two-phase flow conditions. *International Journal of Multiphase Flow* volume 22 issue 1 pp 147-147(1)

[11] Minemura K and Uchiyama T 1993 Three-dimensional Calculation of air-water two-phase flow in centrifugal pump impeller based on a bubbly flow model. *Journal of fluids Engineering* volume 115 issue 4 pp 766-771