Influence of Deviated Short Splitter Vanes on Cavitation Performance of a Low-Specific-Speed Centrifugal Pump

Zikang Li [a], Maoshun Li [b], Xiaobin Gong [c]

Shandong University of Science and Technology, Qingdao 266590, China.

[a]1302741716@qq.com, [b]1101947185 @qq.com, [c]528173250@qq.com

Abstract

On the basis of optimizing the internal flow and external characteristics of the centrifugal pump, improving the cavitation performance has been the main design of the centrifugal pump structure. In order to compare the changes in the cavitation performance of centrifugal pumps at different flow rates, and the effect of short splitter vanes on cavitation performance. Using RNG k-ε turbulence model and Mixture multiphase flow model for the distribution of gas-liquid two-phase inside the impeller of IS50-32-125 centrifugal pump. The analysis results show that the greater the inlet flow of the centrifugal pump, the worse the anti-cavitation performance; The cavitation performance of the centrifugal pump equipped with the offset short blade significantly increased, the maximum bubble volume fraction decreased by 7.3%, 5.9% and 5.3%compared to the prototype pump at flow rates of 10 m^3/h, 12 m^3/h, and 14 m^3/h respectively.

Keywords

Centrifugal pump, deviated short splitter vanes, cavitation performance.

1. Introduction

Low-specific-speed centrifugal pumps are commonly used in agriculture and industrial production. For example, farmland drainage and irrigation, urban and industrial water supply and drainage, thermal power plants, oil refineries, oil pipelines, chemical plants, iron and steel plants, mining companies, ship production and power propulsion, nuclear power plants, rocket fuel supply. However, there are still many problems with centrifugal pumps that have not been solved, and the cavitation problem is one of them. In a centrifugal pump. The pressure of fluid flow is continuously changing, and the lowest pressure point occurs slightly behind the impeller blade inlet. This is where cavitation occurs most easily. Therefore, changing the shape and position of the blade and the structure size of the inlet part of the impeller is the key to improving the cavitation performance of the centrifugal pump.

Most scholars now study the installation of short blades in centrifugal pumps through both structural design and hydraulic characteristics. Qi Xueyi et al. In order to obtain the optimal position of the short blade offset, the analysis and calculation of the addition of the short blade to the blade's back or working face biased the liquid in the leaf turn inside backflow condition. He Youshi et al. understand the flow characteristics in the impeller after adding a short blade by numerical simulation of the centrifugal pump and understand the pressure distribution and velocity distribution in the centrifugal pump. Christopher E. Brennen analyzed the liquid backflow between the centrifugal pump inlet, the impeller, the volute and the internal flow channel, and concluded that the increase in the difficulty of centrifugal pump cavitation was due to the presence of liquid backflow. Philippe Dupont and Tomoyoshi Okamura conducted a comparative analysis of several commercially available CFD (Computational Fluid Dynamics) software for cavitation research. It can be seen that the studies of scholars at home and abroad mainly focus on the influence of geometric characteristics of short blades on the flow field in centrifugal pumps and the theoretical expression of the geometric parameters and external characteristics of long and short blades. So far, the studies on the cavitation of long and short blades have not been common.

This article compares the ordinary centrifugal pump with the rotating speed of 66 and the modified pump under different flow conditions for simulation calculation. Based on the factors affecting the cavitation performance of the centrifugal pump and the bubble volume fraction, bubble distribution and pressure field inside the impeller, the cavitation performance of the centrifugal pump was analyzed to reveal the inside of the pump. The distribution of gas-liquid two-phase flow field provides a theoretical basis for optimizing the cavitation performance of centrifugal pumps.

2. Numerical Calculations

2.1 Geometric Modeling and Meshing

This paper selects IS50-32-125 volute centrifugal pump as the calculation model. The physical model of the prototype pump and the refitting pump with short offset vane are established by the three-dimensional modeling software PRO/Engineer. The entire solid model consists of the volute, the pump inlet extension and the centrifugal pump impeller. Three-dimensional complete model of prototype pump and modified pump. The installed offset short blades are designed according to the following parameters: (1) Inlet diameter D = 0.66D2; (2) Deflection angle α= 0°; (3) Short blade biased to negative pressure side of long blade 0.45θ. Meshing uses Gambit software, uses a tetrahedral unstructured mesh, and uses a node coupling to process the connections between the parts of the impeller that are properly encrypted. Do not number your paper: All manuscripts must be in English, also the table and figure texts, otherwise we cannot publish your paper. Please keep a second copy of your manuscript in your office. When receiving the paper, we assume that the corresponding authors grant us the copyright to use the paper for the book or journal in question. Should authors use tables or figures from other Publications, they must ask the corresponding publishers to grant them the right to publish this material in their paper. Use italic for emphasizing a word or phrase. Do not use boldface typing or capital letters except for section headings (cf. remarks on section headings, below).

Fig 1. Grid model

2.2 Calculation Method and Boundary Conditions Setting

The inlet flow rates were set to 8m³/h, 10m³/h, 12m³/h, 14m³/h, 16m³/h, and 18m³/h, respectively. The model uses the more commonly used RNG K-ε turbulence model and Mixture multiphase flow model. The transmission medium is 25° Cclear water, the temperature is set to 25 °C, the water vaporization pressure is 3170 Pa, the bubble generation coefficient is taken as 50, the bubble collapse coefficient is taken as value 0.01, the standard wall function is selected near the wall surface, and the boundary condition of the wall surface is set to hot non-sliding wall surface, wall roughness is set to 10 μm. The second-order discrete format is used to discretize the calculation region, and the calculation accuracy is set to high precision degree, convergence accuracy is set to 1* 10-5. The inlet boundary condition is set as the pressure inlet, the inlet radius is 25 mm, and the volume fraction of water at the inlet is set to 1, the bubble volume fraction is set to 0. The export boundary condition is set to flow exit. The fluid in the inlet section extension and the volute flow without rotation, in the leaf, the wheel area has a swirling flow, and the interface is set at the exit of the extension zone and the inlet of the impeller, the outlet of the impeller and the inlet of the volute casing, respectively. Non-slip solid Wall boundary conditions, and use the standard wall surface function method to determine the flow near the wall. The surface on the impeller is set as a rotating wall surface, and the other is a fixed wall surface.

3. Numerical Calculation Results and Parameter Analysis

3.1 Pressure Cloud Analysis

The total pressure distribution of the prototype centrifugal pump at a flow rate of 12 m³/h is shown in Fig. 2. The pressure is continuously changed in the centrifugal pump. The flow changes from the inlet to the outlet, and the change in pressure is reduced first. Later at the impeller inlet. The position pressure reaches the lowest point, then gradually increases with the outflow direction, and reaches the maximum pressure when it reaches the outlet. When the absolute pressure of the liquid is lower than the vaporization pressure, bubbles will be generated. When the bubbles are broken by pressure, the cavitation occurs. Therefore, the magnitude of the pressure drop and the size of the low pressure range determine the extent of cavitation.

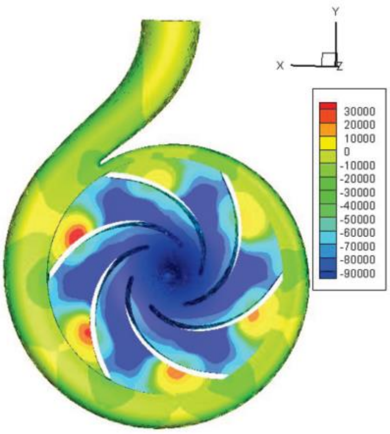

Fig 2. Pressure distribution cloud diagram in flow field at rated flow rate

As the flow rate increases gradually, the low pressure area at the inlet of the pump and the impeller gradually increases and the pressure drop becomes more pronounced. Analyze the reason why the increase in flow means that the inlet speed of the impeller increases, and the hydraulic loss of the water flow in the inlet region of the impeller will increase. At the same time, the flow direction of the water flow into the impeller will be more inconsistent with the orientation of the blades, causing a greater impact loss at the inlet of the impeller. The increased water loss caused by the loss of water pressure causes a partial pressure drop, and when the pressure is reduced to the vaporization pressure, bubbles are generated. Due to the increase of the flow rate, the flow rate of the liquid flows. The speed will also increase, which will cause the bubbles to rapidly spread and cause cavitation to increase.

3.2 Bubble Volume Fraction Analysis

The bubble volume fraction cloud diagram in a centrifugal pump at three different flow rates is shown in Figure 3. When the flow rate is 10 m³/h, the volume fraction of the bubbles is small. At this time, the generation and collapse of the bubbles are in the low pressure area on the back of the blade. When the flow rate increases, the bubbles will rapidly spread due to the acceleration of the centrifugal pump impeller speed and the internal fluid flow rate. Bubbles accumulate in the surface of the blade and in the impeller passage, and a large number of bubbles may also appear at the position of the blade working surface, the impeller outlet, and the like. It can be seen from the figure that the point where the bubble volume fraction is the largest is the center position of the back of the blade, and the bubble volume fraction is smaller along the flow direction closer to the impeller outlet. In these three flow conditions, when the flow rate is 14 m³/h, the bubbles occupy almost the entire impeller flow path, and the cavitation situation is the most serious. After comparative analysis, it was found that within a certain flow range, the bubble volume fraction gradually increased with the increase of the flow rate, but the volume of the bubble distribution region did not change much.

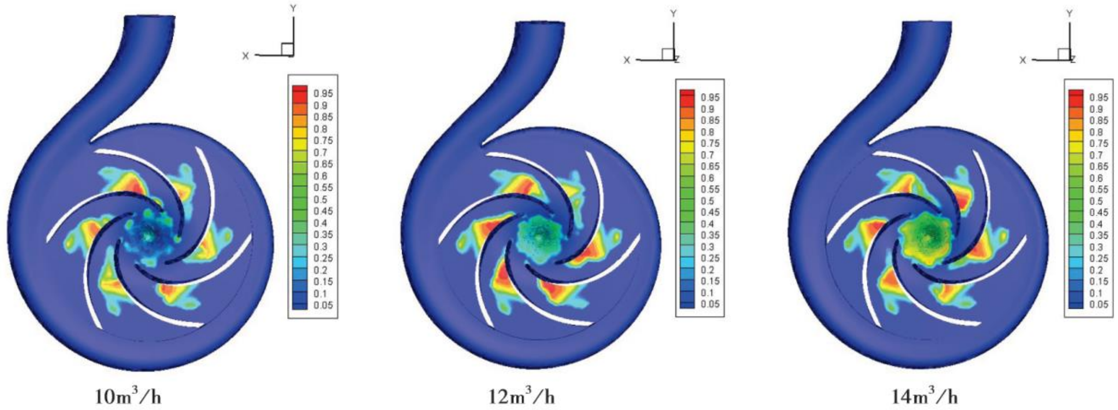

Fig 3. Bubble volume fraction distribution cloud diagram in flow field under different flow rate

In addition, uneven distribution of pressure on the blade surface of the impeller can lead to an asymmetric distribution of air bubbles in the impeller, which is related to the coupling between the impeller and the volute.

3.3 Flow and Impeller Structure Effect on Bubble Volume Fraction

At the same flow rate, the bubble volume fraction of the centrifugal pump equipped with offset short vane is lower than the prototype pump. When the flow rate is 12 m^3/h, the bubble volume fraction decreas by 5.9%; when the flow rate is 14 m^3/h, the bubble volume fraction decreases by 5.3%.

4. Conclusion

(1) When the maximum value of the bubble volume fraction in the centrifugal pump is at the center of the blade's back surface, and then the bubble flows along the impeller flow path to the outlet, the bubble begins to collapse due to the increase of the pressure, and the bubble volume fraction follows the blade. The radial position increases and gradually becomes smaller.

(2) After the centrifugal pump is installed with short offset vanes, the flow conditions inside the prototype pump will be improved. This is because the short vane restrains the irregular flow of some fluids and is the pressure distribution in the impeller and volute. More uniform, to maintain the stability of local pressure.

(3) According to the simulation, in a certain range, the greater the centrifugal pump inlet flow, the worse the anti-cavitation performance. Installation of short offset vanes can effectively suppress the generation or burst of bubbles and improve the anti-cavitation performance of centrifugal pumps.

References

[1] Li X J, Yuan S Q, Pan Z Y, et al. Effects of the near-wall mesh quality on the accuracy of numerical analysis in centrifugal pumps (in Chinese). Trans CSAE, 2012, 28(16): 67-72

[2] Huang B, Wang G Y. Experimental and numerical investigation of unsteady cavitating flows through a 2D hydrofoil. Sci China Tech Sci, 2011, 54(7): 1801-1812.

[3] Mejri I, Bakir F, Rey R, et al. Comparison of computational results obtained from a homogeneous cavitation model with experimental investigations of three inducers. J Fluids Eng-T ASME, 2006, 128(11): 1308-1323.

[4] Bouziad Y A. Physical modeling of leading edge cavitation: computational methodologies and application to hydraulic machinery. Ph.D. Thesis. Lausanne: LMH-EPFL, 2006.

[5] Campous-Amezcua R, Kehelladi S, Bakir F, et al. Numerical analysis of unsteady cavitating flow in an axial inducer. Proc IMech E Part A:J Power Energy, 2010, 224(2): 223-238.

[6] Fredrichs J, Kosna G. Rotating cavitation in a centrifugal pump impeller of low specific speed. J Fluids Eng-T ASME, 2002, 124(6):356-362.