Optimization of centrifugal pump cavitation performance based on CFD

S F Xie, Y Wang1, Z C Liu, Z T Zhu, C Ning and L F Zhao
Research Center of Fluid Machinery Engine and Technology, Jiangsu University, Zhenjiang 212013, China
E-mail: 963489115@qq.com

Abstract. In order to further improve the cavitation performance of a centrifugal pump, slots on impeller blade near inlet were studied and six groups of hydraulic model were designed. Base on cavitating flow feature inside a centrifugal pump, bubble growth and implosion are calculated from the Rayleigh-Plesset equation which describes the dynamic behavior of spherical bubble and RNG κ-ε model was employed to simulate and analyze the internal two-phase flow of the model pump under the same conditions. The simulation results show that slots on blade near inlet could improve the cavitation performance and cavitation performance improvement of the second group was more obvious. Under the same conditions, the pressure on the back of blade near inlet was higher than the pressure on the back of unmodified blade near inlet, and energy distribution in the flow channel between the two blades was more uniform with a small change of head.

1. Introduction
The model pump used in this paper is a constant head pump, which has characteristics such as steady head, high efficiency and so on. Cavitation[1] is a key issue of constant head pump which can cause noise and head fluctuation. In order to further improve the cavitation performance of a centrifugal pump, slots on impeller blade near inlet were studied. Ye Daoxing[2] investigated the effects of slots on impeller blade near inlet and Xing Gang[3] studied the impact of slotted on head. Based on the Reynolds Average Navier-Stokes equation and RNG κ-ε model[4] used in the CFX[5][6] software, this paper simulates internal flow of constant head pump with different impeller under different inlet pressure, and discusses the distribution of vapour volume fraction in impeller. The results could provide theory for improving the cavitation performance of centrifugal pump[7].

2. Models and Boundary Conditions

2.1. Pump geometry mode
The model pump used in present study is a low-specific-speed centrifugal pump with a specific speed of 46.3[8]. Main design parameters of model pump are as follows: \( Q=108 \text{ m}^3/\text{h} \), \( H=140 \text{ m} \), \( n=2970 \text{ r/min} \), impeller suction diameter \( D_1=100 \text{ mm} \), impeller outlet diameter \( D_2=300 \text{ mm} \), impeller width at outlet \( b_2=13 \text{ mm} \), the number of impeller blades \( Z=6 \), blade outlet angle \( \beta_2=90^\circ \). Three-dimensional flow models of different impellers are established by Proe software. In order to ensure uniform distribution

---

1 Correspondence author. Tel.:15751010709.
of the inlet velocity, and to reduce the impact of boundary conditions on flow filed, the sections of the
impeller inlet and volume outlet were carried out to extend , as shown in figure 1. Table 1 shows the
different parameters of the slotted blades , and different impellers as shown in Figure 2.

![Figure 1. Calculation models](image1)

![Figure 2. Different impellers such as table 1](image2)

| Parameter | Number |
|-----------|--------|
| Slot diameter D(mm) | 109 109 109 109 109 109 |
| Width b(mm) | 3 2.65 1.31 1.18 2 1.31 |
| Angle β(°) | 90 0 0 0 90 30 |

2.2. Mesh generation and boundary condition
As is known, performance prediction error will be gradually decreases with the improvement of mesh
quality[9]. In order to get the most economical number of grids and calculate step, this paper identifies
the calculated cell number by the principle of independence of the grid, while structured hexahedral cells
were used to define the inlet and outlet domain, while unstructured tetrahedral cells with strong
flexibility were used for volute and impeller. Finally, the minimum mesh was determined as
1.2mm, and the total number of grid is 1 816 844. The grids number of each domain were 44352,
791495, 856379 and 124600, respectively, as shown in figure 3.

Select of the boundary conditions has a greater impact on the accuracy of the calculation and
results. In this paper, the pump inlet was set as pressure inlet, outlet was set as mass flow. The no-slip
condition for the boundary layers was imposed over walls and the standard wall function was used.
And the interface between impeller and volute was set as frozen-rotor interface, convergence precision
was set to $1 \times 10^{-5}$. 
3. Results analysis

3.1 Vapour volume fraction distribution

Figure 4 shows that the vapour volume fraction distribution of each impeller when the inlet total pressure is set as 101325 Pa. As can be seen from the figure, cavitation is distributed in the back of the blade inlet, and vapour volume fraction of each slotted impeller are reduced compared with the original impeller. This phenomenon of figure (K2) what is that vapour volume fraction is divided into two areas and cavitation occurs in front of the slot is more obvious. Compare with figure 4 (K2), the width of figure 4 (K3) and figure 4 (K4) is reduced while the vapour volume fraction is increase. Compare figure 4 (K3) with figure 4 (K6), it can be seen that a smaller angle can be further reduced vapour volume fraction.

Figure 5 shows that the vapour volume fraction distribution of each impeller when the inlet total pressure is set as 50000 Pa. As can be seen from the figure, compared with the original impeller, vapour volume fraction of each slotted impeller has been significantly reduced, especially K1 and K5. Most of the cavitation area in these two impellers is behind the slot, so the impeller inlet flow channel will not blocked when cavitation occurs. From figure 5 (K1), (K2), (K3), (K4), you can know as the width increases, vapour volume fraction is reduced. Compare figure 5 (K1) with figure 5 (K5), it can be seen that a greater width can be further reduced vapour volume fraction.
3.2 Pressure distribution

Figure 6 shows that the total pressure distribution of each impeller when the inlet pressure is set as 101325 Pa. As can be seen from the figure, it meets the actual phenomenon that total pressure of the back of the blade is lower than the working face, and it can be seen from the figure 6 (Y), (K1), (K5), (K6), there is a low total pressure area in the impeller channel and the area near the back of the blade. As we all know that it means that there is vortex in the impeller channel, it will block the flow channel and reduce hydraulic efficiency. Compare figure 6 (K1) with figure 6 (K5), it can be seen that (K1) has a more significant area of low total pressure what means a smaller width could cause a smaller low total pressure area. This phenomenon has also been confirmed in figure 6 (K1), (K2), (K3), (K4).

Figure 7 shows that the total pressure distribution of each impeller when the inlet pressure is set as 50000 Pa. As can be seen from the figure, The minimum value of total pressure of slotted impeller is increased compared with the original impeller, especially figure 7 (K2) and (K3). While see intuitively from the figure, the total pressure distribution of figure 7 (K2) is better than figure 7 (K3), it has a bigger width than (K3). Compare figure 7 (K1) with figure 7 (K5), it can also be seen that a bigger width could make uniform total pressure distribution in flow channel. Compare figure 7 (K3) with figure 7 (K6), it can be seen that a smaller angle could also cause a uniform total pressure distribution.
In general, a larger width and a smaller angle could make a better total pressure distribution, such as width $b=2.65$ mm, angle $\beta=0^\circ$.

![Figure 7. The total pressure distribution of different impellers under 50000 Pa](image)

3.3 NPSH analysis

Net Positive Suction Head is a very important parameter of the pump performance. There is no accurate calculation method for NPSHR at present except to cavitation experiments. In this paper, numerical simulation of the cavitation experiment had been done by changing the inlet total pressure. Figure 8 shows that NPSH curve of numerical simulation of different impellers.

As can be seen from the figure 8, Slotted impeller has no influence on the head of pump, while NPSH of K2 and K3 have an obvious improvement compared with the original NPSH, they are respectively 0.537 m and 0.42 m. It further evidence that the slotted impeller could improve the cavitation performance of a pump.

![Figure 8. NPSH curve of numerical simulation of different impellers](image)

3.4. Hydraulic efficiency Analysis

Hydraulic efficiency is another important performance parameters of the pump. As can be seen from the table 2, effect of slot on the hydraulic efficiency is little, even the slotted impeller could have a little improvement of hydraulic efficiency of a pump, such as K2, K3.
Table 2. The hydraulic efficiency of the impeller under different inlet pressure

| Inlet pressure | Y     | K1     | K2     | K3     | K4     | K5     | K6     |
|----------------|-------|--------|--------|--------|--------|--------|--------|
| 101325 Pa      | 73.14 | 71.21  | 73.16  | 72.65  | 72.92  | 72.01  | 71.39  |
| 50000 Pa       | 73.01 | 72.35  | 73.01  | 73.11  | 73.02  | 72.83  | 70.71  |
| 20000 Pa       | 41.30 | 61.57  | 67.33  | 67.86  | 59.70  | 59.83  | 61.17  |

4. Conclusions

Based on the above analysis of different parameters and numerical simulation results, some conclusions are obtained:

1) In the above numerical simulation, cavitation is distributed in the back of the blade inlet and it meets the actual phenomenon.
2) Compared with the original impeller, slotted impeller can improve cavitation performance of a low-specific-speed centrifugal pump. While the parameters of slot will be different with different inlet pressure.
3) Compared with the original impeller, slotted impeller could make a more uniform pressure distribution in flow channel, such as width $b=2.65$ mm, angle $\beta=0^\circ$.
4) A reasonable slot with a larger width and smaller angle has no effect on the efficiency of pump.
5) Choose the right slot position, width and angle has a significant impact on the cavitation performance, like near the blade inlet, larger width and a smaller angle.

References

[1] Brennen C E 1994 Cavitation and bubble dynamics (London, Oxford University Press)
[2] Ye D X 2013 Study of impact on the performance of centrifugal pumps with slotted blade (Zhenjiang: Jiangsu University Press)
[3] Xing G 2008 Numerical study of the flow field of centrifugal impeller with Slotted blade (Chongqing: Chongqing University Press)
[4] Launder B E and Spalding D B 1974 Computer Methods in Applied Mechanics and Eng. 3(2) 69-89
[5] Luo X W, Zhang Y, Peng J Q and Xu H Y 2008 J. of Tsinghua University: Sci. and Tech. 48(5) 836-9 (in Chinese)
[6] Wang Y, Liu H, Yuan S, Tan M and Wang K 2011 J. of Drainage and Irrigation Machinery Eng. 29(2) 99-103 (in Chinese)
[7] Wang W J and Wang Y 2012 IOP Conf. Ser.: Earth Environ. Sci. 15(6) 062013
[8] Guan X F 2011 Modern Pump Theory and Design (Beijing: China Astronautic publishing House)
[9] Ferziger J H and Peric M 1996 Computational Methods for Fluid Dynamics (Berlin: Springer)