Influence of blade angle distribution along leading edge on cavitation performance of a centrifugal pump

Y Xu¹, L Tan¹, S L Cao¹,³, Y C Wang¹, G Meng¹,² and W S Qu¹

¹ State Key Laboratory of Hydroscience and Engineering, Tsinghua University, Beijing 100084, China;
² Inner Mongolia Vocational College of Chemical Engineering, Huhehaote 010070, Inner Mongolia, China
E-mail: caoshl@mail.tsinghua.edu.cn

Abstract. The influence of blade angle distribution along leading edge on cavitation performance of centrifugal pumps is analysed in the present paper. Three sets of blade angle distribution along leading edge for three blade inlet angles are chosen to design nine centrifugal pump impellers. The RNG $k$-$\varepsilon$ turbulence model and the Zwart-Gerber-Belamri cavitation model are employed to simulate the cavitation flows in centrifugal pumps with different impellers and the same volute. The numerical results are compared with the experimental data, and the comparison proves that the numerical simulation can accurately predict the cavitation performance of centrifugal pumps. On the basis of the numerical simulations, the pump head variations with pump inlet pressure, and the flow details in centrifugal pump are revealed to demonstrate the influence of blade angle distribution along leading edge on cavitation performances of centrifugal pumps.

1. Introduction
Cavitation is a universal hydrodynamic phenomenon which can lead to the performance degradation and damage to the hydraulic machinery. It happens when the pressure of the liquid is less than the vaporization pressure. In centrifugal pumps, cavitation phenomenon leads to the decrease of the efficiency and impacts the stable operation. Furthermore, the cavity collapse may cause the destruction of the blades and the pump bodies. Due to the strong influence of the cavitation phenomenon on the operation of centrifugal pumps, the investigation of the cavitation flow and its relevant factors was carried out by numerical simulation.

Many studies have been developed to investigate the cavitation flow field in centrifugal pumps. Rudolf et al [1] observed the cavitation at the tongue of the volute and figured out that the 3% head drop may result from the cavitation in the tongue. Tan et al [2] introduced the compressibility into the RNG $k$-$\varepsilon$ model and full cavitation model in order to simulate the cavity accurately at low flow rate. Li et al [3] calculated two pumps with and without the inducer to describe the influence on the leading edge cavitation.

The mechanisms for cavitation have been widely studied, and the relevant parameters of cavitation flow have been investigated. Wang et al [4] figured that the blade incidence angle had a significant impact on the cavitation performance, and there was an optimum value of the angle for cavitation.
characteristic. By using $k$-$\varepsilon$ turbulence model and the volume of fluid cavitation model, Luo et al [5] found that the inlet geometry of impellers had great influence on the cavitation performance.

Previous studies generally indicate that the inlet geometrical parameters of impellers, such as inlet blade angle and inlet edge, have a great influence on the cavitation performance of the pumps. But the effect of the blade angle distribution along leading edge still remains unclear. The main aim of the study is to achieve the influence of blade angle distribution, and to optimize a certain centrifugal pump by changing the distribution rule.

2. Case description and numerical algorithm

2.1. Parameter of the original centrifugal pump

The centrifugal pump is made up of a suction pipe, an impeller with 7 blades and a volute. The main parameters are as follow: flow rate $Q_d=25$ m$^3$/h, pump head $H=7$ m and rotation speed $n=1450$ rpm. The wrap angle of the pump is $98^\circ$ and the diameter of the outlet is 160 mm. The distribution of inlet blade angle along leading edge is shown in Fig.1.

2.2. Distribution ways of inlet blade angle

In order to investigate the influence of the blade angle distribution on pump cavitation performance, nine impellers with the same volute and suction pipe are modelled by changing the blade inlet angle and its distribution way along the blade leading edge. The blade distribution ways of the nine impellers are shown in Table 1. In order to arrange the calculation, the nine pumps are nominated according to the distribution ways of the inlet blade angle.

| Table 1. Blade angle distribution |
|----------------------------------|
| Description | $\beta_1$(hub)/° | $\beta_1$(shroud)/° | $\beta_2$/° |
| S1         | 32             | 46             | 22           |
| S2         | 39             | 39             | 22           |
| S3         | 46             | 32             | 22           |
| M1 (original pump) | 42             | 56             | 22           |
| M2         | 49             | 49             | 22           |
| M3         | 56             | 42             | 22           |
| L1         | 52             | 66             | 22           |
| L2         | 59             | 59             | 22           |
| L3         | 66             | 52             | 22           |

Figure 1. Distribution of blade angle along blade leading edge of the original pump.
2.3. Computational Methodology

The numerical analysis is carried out by the CFD software ANSYS-CFX. The RNG $k$-$\varepsilon$ turbulence model with scalable wall functions is employed to calculate the turbulent flow. The cavitation model is Zwart-Gerber-Belamri model based on Rayleigh-Plesset Equation, which is widely used in predicting the inception, growth and collapse of the bubbles. The convergence condition is set to $1.0 \times 10^{-5}$ of the RMS residual. For the non-cavitation regime, the boundary conditions are assumed: at the inlet of the domain the pressure is specified, and at the outlet, the mass flow rate is defined. Then for the cavitation condition, the inlet pressure decreases.

3. Experimental and numerical results of the centrifugal pumps

3.1. Experimental results and numerical simulation of the original pump

The experiment is conducted on an open-architectured platform which meets the national standard on centrifugal pump experiment. The electromagnetic flowmeter is used to measure the flow rate. The head of the pump is determined by the total pressure at the inlet and outlet according to the function (1) where $p_1$ and $p_2$ is the pressure of inlet and outlet of the pump; $\rho$ is the fluid density; $Z$ is the height of the inlet and outlet; $v_1$ and $v_2$ is the velocity of the fluid in the inlet and outlet. The efficiency is defined as function (2) where the $H_t$ is the theoretical head value of the pump.

$$
H = \frac{p_2 - p_1}{\rho g} + (Z_2 - Z_1) + \frac{v_2^2 - v_1^2}{2g} 
$$

$$
\eta = \frac{H}{H_t} \times 100\%
$$

While testing the head and efficiency, the cavitation performance of the original pump is tested by reducing the inlet pressure.

In order to insure the accuracy of numerical prediction, the performance of the original pump is calculated and compared with the experiment data. The inlet and outlet domains are extended to reduce the effect of boundary conditions. The ANSYS-TURBOGRID is used to generate the hexagonal structured grids of impeller and the hexagonal grids of volute and suction pipe are generated by ICEM. Then the grid dependence analysis is carried out, the details and the results of the four grids are in the Table 2. The three finest grids present the results that are in good agreement with the data of experiment. Grid 2 is selected for this study, and the topology is used to generate all the grids of calculations. Figure 2 shows the mesh of the impeller.

| Grid No. | Total grid size | Efficiency (%) | Head(m) |
|----------|-----------------|----------------|---------|
| Grid 1   | 944962          | 87.21          | 7.15    |
| Grid 2   | 1110652         | 88.41          | 7.17    |
| Grid 3   | 1169011         | 88.53          | 7.17    |
| Grid 4   | 1246837         | 88.49          | 7.17    |
The hydraulic performance and cavitation performance are calculated. The results of hydraulic performance and the comparison with experimental data is shown in Figure 3. The cavitation curves are displayed in Figure 4, and the values of NPSH of different flow rate are compared with the data of experiment in Figure 5. At each flow rate, the numerical simulation presented a well prediction. The results indicated that the numerical simulation agreed with the experimental results, so the results are reliable.

**Figure 2.** Mesh of impeller

**Figure 3.** Comparison of hydraulic performance.

**Figure 4.** Cavitation curves for different flow rates.

**Figure 5.** Comparison of cavitation performance
4. Cavitation performance of centrifugal pumps with different blade distributions

For modelling the nine impellers according to the distribution of blade inlet angle, Bladegen is employed to generate the three-dimension model of the impellers.

In order to calculate the cavitation performance of the pumps, the performance under non-cavitation regime are calculated first. Taking the results under non-cavitation condition as the initial conditions, the cavitation performances are calculated. As reducing the value of inlet total pressure, the value of $\text{NPSH}_c$ decreased and the cavitation curves are obtained.

The value of $\text{NPSH}_c$ is chosen as the evaluation criterion of the performance of cavitation. The values are obtained from the cavitation curves and are shown in Table 3 and the comparison of the nine pumps is shown in Figure 6.

It is obvious that the average value of the inlet blade angle along leading edge has significant influence on the cavitation performance of the pump. The best value is 49°, and its average value of $\text{NPSH}_c$ is 1.586m which is lower than the value of 39° for 0.05m and the value of 59° for 0.064m.

The results of three average values of inlet angle show the same conclusion that the evenly distribution way is the best distribution for cavitation performance. And the distribution ways of mounting and tapered show performance similarly. So it is helpful for optimizing cavitation performance to design pumps with evenly distribution of inlet blade angle along the leading edge.

| Pump No.  | $\text{NPSH}_c$ (m) |
|-----------|---------------------|
| S1        | 1.643               |
| S2        | 1.620               |
| S3        | 1.644               |
| M1 (original pump) | 1.604           |
| M2        | 1.575               |
| M3        | 1.580               |
| L1        | 1.655               |
| L2        | 1.640               |
| L3        | 1.655               |

Figure 6. $\text{NPSH}_c$ of the pumps

5. Cavitation performance of the best pump

5.1. Cavitation process of the best performance pump

The analysis results indicate that the pump M2 has the best cavitation performance. In order to analyse the inception, development and collapse of cavitation, the inner flow fields of pump M2 under
different NPSH, value are investigated. It is found that the cavitation occurs in the inlet of suction side of the blade first. As decreasing the value of total pressure at inlet, the range of cavitation expands to the middle of the suction pipe, and finally occupies the passage, which reduces the efficiency and head of the pump. When the value of NPSH reduces to 1.07m, part of the bubble separate from the primary flow and run into the volute. The whole process of cavitation is shown in Figure 7.

![Figure 7. Cavitation process of the best performance pump M2](image)

5.2. Blade load

Figure 8 shows the static pressure distributions along the blade under different value of NPSH. At decreasing the value of NPSH, the net pressure got smaller. When the value reach to 1.27m, the trend of the pressure distribution changes, and the suction effect near the leading edge reduces. The influence is significant in suction side, and the pressure mainly reduces near the inlet. When the value further reduces to 1.07m, the value of pressure decreases in both two sides near the leading edge. The distribution of the pressure coincides with the cavitation process.

![Figure 8. Static pressure distributions under different cavitation condition.](image)
6. Conclusions
The RNG $k-\varepsilon$ turbulence model and Zwart-Gerber-Belamri cavitation model are employed to predict the cavitation performance of nine pumps with different distributions of blade inlet angle. The values of $NPSH_r$ of the nine pumps are obtained by composing the cavitation curves. The inner flow and blade loading under different cavitation condition of the best performance pump are also analyzed. The main conclusion are as follows:

1) The best distribution way is evenly way and the best average value of the inlet blade angle is $49^\circ$. The other two ways perform similarly.

2) The cavitation first occurs near the leading edge of suction side at impeller hub, then expand to the whole passage. It is also notable that the range of cavitation moved from hub to shroud while decreasing the value of $NPSH_r$. The inlet geometry has decisive influence on the cavitation performance. To ensure the flow uniformity of the inlet by selecting proper inlet parameters is an effective way for reducing the cavitation in pumps.

References
[1] Bachert R, Stoffel B and Dular M 2010 J. of Fluids Eng. 132(6) 061301.
[2] Tan L, Zhu B, Cao S and Wang Y 2013 Chinese Sci. Bulletin 58(8) 949-52.
[3] Li X J, Pan Z Y, Zhang D Q and Yuan S Q 2012 Pro. of the 26th IAHR Sym. on Hydraulic Machinery and Systems(Beijing, China, 19-23, August, 2012)
[4] Wang Y, Liu H L, Yuan S Q, Tan M G and Wang K 2011 Fluid Machinery 4 005
[5] Luo X W, Zhang Y, Peng J Q and Xu H Y 2008 J. Tsinghua Univ. (Sci. & Techno. ) 05