Numerical research on the cavitation characteristics for typical conditions of a centrifugal pump with whole flow passage

L Cao, Z W Wang, Y X Xiao, Y Y Yao and W Zhu

State Key Laboratory of Hydroscience and Engineering & Department of Thermal Engineering, Tsinghua University, Haidian District, Beijing 100084, China

E-mail: wzw@mail.tsinghua.edu.cn

Abstract. Cavitation is one of the key issues affecting the safe and stable operation of centrifugal pumps. This research conducted numerical simulations of the 3-D turbulent flow in the whole flow passage of a centrifugal pump using RANS method. The calculation results of cavitation characteristics agreed well with the experimental results, which were based on SST $k-\omega$ turbulence model and Zwart-Gerber-Belamri cavitation model. This paper analysed the cavitation development process and the corresponding pump performance for three typical conditions, namely large discharge condition, design discharge condition and small discharge condition, by changing the available Net Positive Suction Head ($NPSHa$). For large discharge condition, the incipient $NPSHa$ was large, while for design discharge condition and small discharge condition, the incipient $NPSHa$ values were almost the same and both small. As the flow rate decreased, the critical $NPSHa$ decreased as well, and the cavitation position shifted from the pressure surfaces of some blades to the suction surfaces. At the same time, the tongue has greater effect with larger flow rate and the cavitation becomes less unsteady with the decrease of flow rate. With similar vapour volume, cavitation on the blade pressure side more easily leads to the drop of pump performance. Therefore, more attention should be paid to the cavitation characteristics of centrifugal pumps in large flow conditions in hydraulic design stage.

1. Introduction

Since cavitation is one of the critical problems that restrict the promotion of pump performance, cavitation mechanism and control measures for centrifugal pumps are researching focuses all the time. Cavitation characteristics are not the same in different flow rates, and different cavitation condition will affect the pump performance to different degree. It is meaningful to probe into the cavitation characteristics in different flow rates and make out the mechanism that how cavitation affects pump performance for sake of better hydraulic design and safe and stable operation of centrifugal pumps.

Detailed information of the 3-D cavitating flows in centrifugal pumps can be conveniently obtained by numerical simulations, which is conductive to carry out researches on flow mechanism. Due to the complexity of cavitation, precise prediction of the cavitation characteristics of centrifugal pumps is always difficult. In recent years, there have been many research achievements on cavitation numerical...
methods and cavitation models for centrifugal pumps. Medvitz et al. conducted quasi-3-D calculation and analysis of cavitation developments in several flow rates for a centrifugal pump and explain the issue of cavitation breakdown with the corresponding flow structure. Dijkers et al. accurately calculated the sheet cavitation inside the centrifugal impeller with a potential-flow model. Pouffary et al. did a quick and precise prediction on the 3-D cavitation features in a centrifugal pump and thought that it was the variation of blade load that gave rise to cavitation breakdown. Wang et al. calculated the cavitating flows in three centrifugal pumps of different specific speed by neural network method and CFD numerical simulation method respectively, and presented that neural network method could be applied in the optimal design of centrifugal pumps since it was convenient to control the relation between input parameters and output parameters. As for turbulence models in cavitation calculations, several literatures modified the RNG $k$-ε turbulence model by using density function to correct the turbulent viscosity item. In addition, some literatures proposed that cavitation model should take turbulent pressure pulsation into consideration to correct the phase-change pressure. These efforts improved the prediction accuracy of cavitation characteristics to some extent and captured the cavitation features in different operation conditions of centrifugal pumps.

Up to now, explanations for mechanism of cavitation affecting pump performance mainly concentrated on two aspects. One is that different cavitation coefficients will induce cavitation of different volumes and positions, which influences the impeller discharge capacity at different levels and change the flow structure inside the impeller passage as well. The other is that cavitation will change the load distribution on the blades. However, the pump performance change law still remains unexplained in some conditions. Recently, clearance flows in the side chambers between impeller shrouds and casing covers are usually ignored in numerical simulations. Constantin et al., Li et al., Si et al. proposed that the unsteady flows in the side chambers should be considered in order to get a more accurate cavitation calculation results.

In this article, numerical research on the cavitation characteristics in thre typical conditions of a shrouded centrifugal pump is carried out by cavitation calculation for the whole flow passage considering clearance flow. Cavitation characteristic curves and vapour distribution laws in these conditions are obtained and comparatively analysed, which is meaningful to reveal the mechanism that how cavitation affects the pump performance.

2. Physical model and numerical methodology

2.1. Geometry model and computational grid
The centrifugal pump in this article has a three-blade shrouded impeller whose outlet diameter $D_2$ is 310 mm. The specific speed $n_s$ is 165. There are nine backward curved back blades on the front shroud. 3-D model of the whole flow passage was set up by software UG as shown in figure 1 and the geometry was separated to six domains including suction pipe, impeller, volute casing, back-blade region, front chamber and back chamber. The minimum axial dimension of the front chamber is 0.22 mm, and the minimum axial dimension of the back chamber is 0.78 mm.

Discretize the impeller domain with unstructured grids so that the element size at the interfaces between impeller and side chambers could be better controlled. Other domains were discretized by structured grid. Four sets of grids having 0.9 million to 2.0 million elements were used in calculation in design condition. The result with 1.63 million or more elements had no significant difference of less than 0.2%. Thus, the grid with 1.63 million elements was used to balance the calculation accuracy and the cost. The grid for calculation is shown in figure 2 and the detailed information is listed in table 1.

2.2. Numerical model and computational method
The governing equations were solved by commercial CFD software Ansys CFX 14.5 with RANS method. Apply homogeneous multiphase model to calculate the liquid-vapour two-phase flow and use Reyleigh-Plesset model to deal with mass transport between water and vapour at 25 °C. The saturated vapour pressure at 25 °C is 3574 Pa. In order to give a better prediction of flow separation and
rotational flow, use SST k-ω turbulence model to close the governing equations. Pressure field and velocity field were coupled by SIMPLEC algorithm with high resolution scheme. Calculations were thought to get to convergence until the residual mean squares decreased to 0.0001.

**Figure 1.** 3D model of whole flow passage of the dredging pump.

**Figure 2.** Grid.

**Table 1.** Detailed mesh information.

| Domains                  | Suction Pipe | Impeller | Volute Casing | Back-blade region | Front Chamber | Back Chamber | Total |
|--------------------------|--------------|----------|---------------|-------------------|---------------|--------------|-------|
| Number of nodes          | 70           | 130      | 680           | 50                | 90            | 150          | 1170  |
| ($\times 10^3$)          |              |          |               |                   |               |              |       |
| Number of elements       | 70           | 670      | 650           | 40                | 80            | 120          | 1630  |
| ($\times 10^3$)          |              |          |               |                   |               |              |       |

Give total pressure values at the inlet of the suction pipe and give average velocity at the outlet of the volute casing. The impeller domain and back-blade region were set as rotational domains with a rotating speed $n=1000$ r/min, and other domains were stationary. Frozen rotor model was used at the interfaces between rotational domain and stationary domains. The walls in side chambers, adjacent to the outside surfaces of the front shroud and the back shroud, were set as rotational walls with the same rotating speed as that of the impeller, and other walls were all set as no-slip walls. The reference pressure was set as 0 Pa. For each flow rate, calculation began from single-phase flow with high total pressure at the inlet. Gradually reduce the total pressure at the inlet. When the local pressure was lower than the saturated vapour pressure, add cavitation model to conduct two-phase flow calculation. Keep reduce the total pressure at the inlet until cavitation was seriously developed. In this paper, cavitation calculations were carried out for three flow rates, namely $1.3Q_{th}$, $Q_{th}$ and $0.625Q_{th}$, where $Q_{th}$ is the design flow rate. Considering the asymmetry of the volute casing and the blade-tongue interaction, each condition was calculated three times with the impeller rotated 40°each time and the arithmetic mean values of these three results were then used as the final pump performance parameters for the corresponding conditions. In addition, in order to illustrate the influence of clearance flow on the cavitation characteristics, cavitating flows were predicted without side chambers in flow rate of 1.3 $Q_{th}$ as well.

3. Results and discussion

3.1. Validation of numerical results
The cavitation tests were conducted in a closed test rig with a measurement uncertainty of less than ±0.2% and the pump rotating speed is 1000 r/min. With flow rate of $1.3Q_{th}$, the first test point was in non-cavitation condition. Then gradually increase the vacuum degree of the pump system by a vacuum pump to reduce the available Net Positive Suction Head ($NPSHa$) until the efficiency or head dropped to a certain degree. Based on the head of the initial point, $H_0$, calculate the increment or decrement of the head of other conditions, $\Delta H$, and then obtain the $\Delta H/H_0$-$NPSHa$ curve. Cavitation characteristic curves in $1.3Q_{th}$ obtained experimentally and numerically are both given in figure 3. The black curve represents the experimental result, while the red curve represents the numerical result with side chambers and the blue curve represents the numerical result without side chambers. It can be seen from the figure that the result considering clearance flow agrees better with the experimental result.

![Figure 3. Comparison of experimental and numerical $H$-$NPSHa$ curves.](image)

3.2. Cavitation characteristics in different flow rates

It is still difficult to measure the incipient cavitation coefficient by experiment at present, while numerical simulation can conveniently predict the incipient cavitation. Figure 4 gives the cavitation characteristic curves of this pump in $1.3Q_{th}$, $Q_{th}$ and $0.625Q_{th}$. In $1.3Q_{th}$, cavitation has occurred when $NPSHa=9$ m, while the $NPSHa$ values are much smaller for $Q_{th}$ and $0.625Q_{th}$ which is about 5 m. Regard the operation point where the head dropped by 3% as the critical cavitation point. The critical $NPSHa$ decreases as the flow rate decreases. This result is consistent with the previous research findings that cavitation is apt to occur in large flow rates and has more significant effect on the pump performance.

![Figure 4. $H$-$NPSHa$ curves for different flow rates.](image)
In different flow rates, the cavitation position and its evolution process is also different. Figure 5 to figure 7 respectively show the incipient cavitation, developing cavitation and critical cavitation for the three flow rates. The blue regions represent the iso-surface of vapour volume fraction $\alpha=0.1$. The walls of impeller are set as semi-transparent. So in the given orientation, vapour on the pressure surface presents faint blue due to cover of impeller walls, and the uncovered vapour presents brilliant blue.

From figure 5, in the large flow rate, cavitation first occurs on the pressure surface of one blade and near the front shroud side at the blade inlet. It is easy to cavitate at the blade leading edge which is close to the tongue. As the impeller rotates, cavitation alternately occurs at inlet of different blades. In the design flow rate, the incipient cavitation position is still related to the blade-tongue relative position. Cavitation appears on the pressure surface of the blade whose leading edge is near the tongue and at the same time appears on the suction surface of the blade whose leading edge is far away from the tongue. In the small flow rate, the incipient cavitation completely occurs on the suction surface of the blade whose trailing edge is near the tongue. As to the reason of these phenomena, it is negative attack angle at the impeller inlet for large flow rate conditions and the incoming flow impacts the suction surface of the blade which may cause flow separation at the pressure surface and induce cavitation. On the contrary, in small flow rate conditions, it is positive attack angle at the impeller inlet and the incoming flow impacts the pressure surface of the blade which may cause flow separation at the suction surface and induce cavitation. Furthermore, cavitation feature is different for the three impeller angular positions which indicates that the incipient cavitation is unsteady.

![Figure 5. Iso-surfaces of vapour volume fraction $\alpha=0.1$ in incipient cavitation conditions.](image-url)
As the inlet total pressure falls, vapour volume increases. Figure 6 shows the developing cavitation conditions when the pump performance has not changed dramatically. For $1.3Q_{th}$, vapour still concentrates at the front shroud side of the leading edge of some blades and great influence of the tongue still exists, and cavitation has extended to the inside surface of the front shroud. Whereas for $Q_{th}$, the vapour mainly concentrates on the suction surface of the blades and the tongue effect has weakened. Cavitation extends from the front shroud side towards the back shroud side and occurs on the inside surface of the front shroud as well. For $0.625Q_{th}$, cavitation extends from the front shroud side towards the back shroud side similarly and vapour distribute evenly on the suction surfaces of the three blades which is hardly influenced by the tongue. Therefore, cavitation becomes less unsteady with the decrease of flow rate.

![Figure 6. Isosurfaces of vapor volume fraction $\alpha=0.1$ in developing cavitation conditions.](image)

Continue to reduce the inlet total pressure, when cavitation develops to a certain extent, efficiency and head of the pump will drop remarkably. Figure 7 shows vapour distribution around the critical cavitation point. Vapour volume goes on increasing for each flow rate. The effect of tongue remains in the large flow rate condition and cavitation has develops to the suction surface. In the design condition and small flow rate condition, cavitation both develops towards the blade outlet. The increment of vapour volume is so large that has caused obvious change in the flow structure inside the blade channel, especially near the leading edge, and lead to deterioration of the pump performance.
It is worth noting that the vapor volume is similar for conditions shown in figure 7(a) and figure 6(b) but there is a big difference in their influence on pump performance. Since the blade works depending on the pressure surface, so cavitation on the pressure surface in large flow rate conditions affects the pump performance much greatly than that occurring on the suction side with less flow rate. Therefore, more attention should be paid to cavitation characteristics of centrifugal pumps in large flow conditions in hydraulic design stage.

![Isosurfaces of vapor volume fraction $\alpha=0.1$ in critical cavitation conditions](image)

**Figure 7.** Isosurfaces of vapor volume fraction $\alpha=0.1$ in critical cavitation conditions.

### 4. Conclusion

This article studies the cavitation evolution law and the effect of cavitation on pump performance via numerical simulations of cavitating flows in three typical flow rates. Compared with experimental results, it finds that considering the clearance flow can get a more accurate cavitation characteristic curve. Through comparative analysis of the cavitation characteristics for the three flow rates, it shows that cavitation is apt to occur in large flow rate conditions and meanwhile the tongue has greater effect with larger flow rate. As the flow rate decreases, the critical $NPSHa$ decreases and the cavitation positions gradually shift from the pressure surfaces of some blades to the suction surfaces. With similar vapor volume, cavitation occurring on the blade pressure side is easier to lead to pump performance drop. Therefore, more attention should be paid to the cavitation characteristics of centrifugal pumps in large flow conditions in hydraulic design stage.
Acknowledgments
Special thanks are due to the State Key Program of National Science of China (Grant No. 51439002), National Natural Science Foundation of China under Contract (No. 51479093), State Key Laboratory of Hydrosience and Engineering (Grant No. 2014-KY-05) for supporting the present work.

References
[1] Medvitz R B, Kunz R F, Boger D A, Lindau J W, Yocum A M and Pauley L L 2002 J. Fluid Eng. 124(2) 377-383
[2] Dijkers R J H, Fumex B, Woerd J G H O, Kruyt N P and Hoeijmakers H W M 2005 Proc. ASME 2005 Fluids Engineering Division Summer Meeting (Houston, Texas, USA, 19-23 June 2005) pp 1233-8
[3] Pouffary B, Patella R F, Reboud J L and Lambert P A 2008 J. Fluid Eng. 130(6) 061301
[4] Wang Y, Liu H L, Yuan S Q, Tan M G and Wang K 2009 Proc. the IEEE Int. Conf. on Computing and Intelligent Systems 1 137-40
[5] Liu H L, Liu D X, Wang Y and Zhuang S G 2012 J. Huazhong Univ. of Sci. & Tech. (Natural Science Edi.) 40(8) 17-20
[6] Tan L, Cao S L, Wang Y M and Zhu B S 2012 J. Chin. Phys. Lett. 29(1) 014702
[7] Tan L, Zhu B S, Cao S L and Wang Y M 2013 J. Chin. Sci. Bull. 58(8) 949-52
[8] Liu H, Liu D X, Wang Y, Wu X F, Wang J and Du H 2013 J. Hydrodynamics B 25(3) 370-8
[9] Wang W J, Wang Y, Liu R H, Li Y B, Yin G and Li G D 2014 T. Chin. Soc. Agr. Mach. 45(3) 37-44
[10] Li J, Liu L J and Feng Z P 2006 J. Xi’an Jiaotong Univ. 40(3) 257-60
[11] Wang X L, Yuan S Q, Zhu R S, Fu Q and Yu Z S 2012 T. Chin. Soc. Agr. Mach. 43(3) 67-72
[12] Yuan J, Fu Y and Yuan S 2012 ASME 2012 Fluids Engineering Division Summer Meeting colocated with the ASME 2012 Heat Transfer Summer Conference and the ASME 2012 10th International Conference on Nanochannels, Microchannels and Minichannels (Rio Grande, Puerto Rico, USA, 8-12 July 2012) pp 193-202
[13] Luo X, Wei W, Ji B, Pan Z Zhou W and Xu H 2013 J. Mech. Sci. Tech. 27 (6) 1643-48
[14] Li X J, Yuan S Q, Pan Z Y, Yuan J P and Fu Y X 2013 J. Sci. Chin. Tech. Sci. 56 (9) 2156-62
[15] Wang Y, Liu H L, Yuan S Q, Tan M G and Wang K 2011 J. Drain. Irrig. Mach. Eng. 29 (2) 99-103
[16] He M, Fu L P, Zhou L J, Guo Q and Wang Z W 2012 The 26th IAHR Symposium on Hydraulic Machinery and Systems (Beijing, China, 19-23 Aug 2012) 15(6) 062022
[17] Constantin S L 2012 Int. Society for Optics and Photonics 841131-10
[18] Si Q, Yuan S, Li X, Yuan J and Lu J 2014 T. Chin. Soc. Agr. Mach. 45(5) 84-90