Numerical calculation for cavitation flow of inducer

C Ning, Y Wang, Z T Zhu, S F Xie, L F Zhao and Z C Liu
Research Center of Fluid Machinery Engineering and Technology, Jiangsu University, Zhenjiang 212013, China

E-mail: dream0830@163.com

Abstract: Inducer has significant effect on improving the cavitation characteristic of centrifugal pump. Several inducers were designed and modeled by Pro/E software. The mesh of flow field was done by ICEM and then was imported to ANSYS CFX to analyze the inducer’s cavitation characteristic. Effects of the blade number on the performance of an inducer are investigated in the present paper. The inducers were designed on the basis of identical design flow rate and identical pressure elevation at nominal flow rate. The study focuses on the steady behavior of the inducers in cavitating conditions. Evolutions of performance, torque, mass flow rate, and amplitude of radial forces on the shaft according to the inlet pressure are considered. Furthermore, cavitation instabilities are analyzed in the study. The purpose of the present study is to investigate the pressure distribution and vapour volume fraction distribution through numerical simulations using the Navier-stokes solver with computational fluid dynamics (CFD) code.

1. Introduction

The phenomenon of cavitation in centrifugal pump is an important feature of the process of its operation and one of the concern issues of the design, manufacture, use and management of a centrifugal pump.

In the process of pump operation, for some reason, when the absolute pressure of conveying liquid medium dropped to vaporization pressure at that temperature in the flow section of the local area, liquid in this area begin to boil, produce steam, form bubbles. These bubbles flow forward along with the liquid until a certain place of high pressure, high pressure liquid around the bubbles result in the bubbles shrink sharply that broken, and at the same time, the liquid particle filling holes at high speed, hit each other and form water hammer. This phenomenon will make the flow components suffer from corrosion damage when occurred in the solid wall. The process which bubbles was formed and then ruptured and cause damage to flow components is the process of water pump cavitation.

Inducer is actually a kind of axial flow impeller with small blade load, the effect of the centrifugal force is small. It is not easy to produce the two phase separation of liquid and bubbles. It has high performance of resistance to cavitation. It also can work normally and not blocked even under the condition of local cavitation. For centrifugal pump, the model of installing inducer in front of the centrifugal impeller, the essence of which is greatly reduces the required NPSH of pump unit, was used more in order to improve its performance of resistance to cavitation. In order to improve both the cavitation resistance of the inducer itself and the head of the inducer to meet the requirement of high cavitation performance of the main impeller, the model of varying pitch inducer is often used. Under the condition of same impeller import outer diameter, the variable pitch inducer can more effectively reduce the necessary NPSH of pump unit than uniform pitch inducer.
Perfect design theory of inducer is summarized in the early 70s by NASA (National Aeronautics & Space Administration) and the former Soviet union respectively according to their work in the field of aerospace. After more than 20 years, the design theory and the development effort of inducer in our country is basically on the basis of the job.

Huang Jiande \(^1\), further tested four different kinds of inducers on blade number and blade length, and find out the influence of the above phenomenon happened and the cavitation performance by impeller geometry, and made a preliminary study on the reason.

Zhu Zuchao, etc. \(^3\), established a relatively perfect design method of varying pitch inducer through the analysis of the theoretical basis of inducer design, the calculation formula of its main parameters was given, the calculation method of cavitation performance and theoretical head of the inducer was deduced \(^3\).

Sun Jian \(^4\), put forward that both the inducer itself no cavitation and the export head meet the requirements of import energy of main impeller of the centrifugal pump were the design principles, and further expounded the structure matching relationship between the inducer and the centrifugal pump impeller.

In this study, cavitation instabilities are analyzed. Effects of the blade number on the performance of an inducer are investigated in the present paper.

2. Design and modeling of the centrifugal pump and inducers

Selecting a centrifugal pump with the specific speed is 103 to numerical simulation and analysis of the design flow rate is 660 m\(^3\)/h, head is 67 m, the speed is 1450 r/min, import a diameter of 250 mm, entity modeling in Pro/E, in order to avoid impact on the calculation results of velocity grads, import and export of the spiral case was extended appropriately.

To match the pump, the flow rate of inducer is 660 m\(^3\)/h, head is 49 m, speed is 1450r/min. NPSH is 1.2 m. As shown in figure 1.

![Figure 1. Three-dimensional model of inducers](image)

3. Calculation method of cavitation flow

3.1. Meshing

The leakage of the tip clearance and the aroused backflow should be analyzed, so when the water
model was created, a tip clearance of 1 mm was considered. Water model was meshed after imported to ANSYS-ICEM software. Due to the structure of inducer is relatively complex, adaptive unstructured grid method was used. Because the tip clearance was tiny, grids near the tip were refined. Cavitation mainly happens near the blade surface, so in order to accurately simulate the cavitation development, grids near the blade surface were also refined, and the quality of grids were above 0.3.

3.2. Calculation method
For most of cavitation flow simulation, the typical homogeneous flow model in ANSYS CFX was used. The mass transfer process between the liquid phase and vapor phase was calculated by Rayleigh-Plesset equation\(^5\).

Continuity equation and the Reynolds averaged Navier-Stokes equations were used to simulate the flow field in the inducer, and the RNG \(k-\varepsilon\) turbulence model was used to close the equations.

The total pressure of inlet and the mass flow of outlet were selected as the boundary condition. By adjusting the inlet total pressure, the development of inducer’s cavitation was controlled. The initial conditions of the cavitation calculation were depended on the results of non-cavitation results. The 25°C water was chosen as the medium, and the vapor pressure was set to be 3169 Pa. The average diameter of bubbles was set to be \(2.0 \times 10^{-6}\) m. The frozen-rotor interface was selected as the interface between inducer and the inlet or outlet. The convergence precision was set to be \(1.0 \times 10^{-5}\).

4. Results analysis

4.1. Pressure distribution when the medium is water
Figure 2 shows that the total pressure distribution of each vane when the inlet pressure is set as 40000 Pa. It can be seen from the figure 2 (1), (2), (4), there is a low pressure area in the impeller channel. Compare figure 2 (1) with figure 3 (2), (3), (4), you can see that (1) has a more significant area of low pressure. And (3) has a less significant area of low pressure.

4.2 Pressure distribution when the mediums are water and vapour
Figure 3 shows that the total pressure distribution of each impeller when the mediums are water and vapour.

4.3 Vapour volume fraction distribution when the mediums are water and vapour
Figure 4 shows that the vapour volume fraction distribution of each vane when the inlet total pressure is set as 40000 Pa. As can be seen from the figure, cavitation is distributed in the inlet of blade, and vapour volume fraction of each vane in figure 4(2) are reduced compared with the original impeller. This phenomenon of figure 4(2) is that vapour volume fraction is disappear and cavitation occurs in some area of the back of the impeller.

4.4 Vapour volume fraction distribution under 30000 Pa
In order to magnify the effect of cavitation, the inlet total pressure is set as 30000 Pa.

Figure 5 shows that the vapour volume fraction distribution of each vane when the inlet total pressure is set as 30000 Pa. As can be seen from the figure, the range of cavitation on the impeller is small, but the location of cavitation on the blades of inducer is different. With the increasing of the blade number of inducer, cavitation begins to occur on the back of the blades.
Figure 2. Pressure distribution under 40000Pa
5. Conclusions

1) The inducer has influence on cavitation performance of centrifugal pump, can greatly improve the cavitation performance of the pump.

2) With the increase of the blade number of inducer, the pressure distribution become non-uniform, so the even distribution of relative velocity is non-uniform. In order to prevent the alternate blade cavitation, hope blade for the odd number.

3) The blade number of main impeller is better equal to multiple the blade number of inducer, the flow which contains certain rotating components after come out from the inducer enters the primary impeller, thus the angle of fluid flow which enters the primary impeller is increasing.
(1) with inducer of 2 blades

(2) with inducer of 3 blades

(3) with inducer of 4 blades

Figure 5. Vapour volume fraction distribution under 30000Pa

Acknowledgment
The authors gratefully acknowledge the financial support of National Science and technology Support Plan of China (2011BAF14B01), a project funded by the Priority Academic Program Development of Jiangsu Higher Education Institutions (BK2009218). Conflict of interests: The authors declare no conflicts of interest. The support of National Science and technology Support Plan of China (2011BAF14B01), a Project Funded by the Priority Academic Program Development of Jiangsu Higher Education Institutions (BK2009218) have no conflict of interest.

References
[1] Huang J D, Gu C G 2000 J. of Aerospace Power 2000(6) 142-6 (in Chinese)
[2] Zhu Z C, Wang X R and Wang L Q 1997 J. of industry machinery 1997(4) 102-4
[3] Zhu Z C and Wang Y Q 1998 J. of Zhejiang University 1998(3) 197-200
[4] Sun J, Kong F Y and Jiao Q B 2006 J. of fluid machinery 2006(3) 19-24
[5] Mejri I, Bakir F and Kouidri S 2006 J. Power Energy 220 217