Cavitation performance evaluation for a condensate pump

A Yu\textsuperscript{1,2}, W P Yu\textsuperscript{3}, Z B Pan\textsuperscript{4}, X W Luo\textsuperscript{1,2}, B Ji\textsuperscript{1} and H Y Xu\textsuperscript{1}

\textsuperscript{1} State Key Laboratory of Hydrosience and Engineering, Tsinghua University, Beijing, 100084, China
\textsuperscript{2} Beijing Key Laboratory of CO\textsubscript{2} Utilization and Reduction Technology, Tsinghua University, Beijing, 100084, China
\textsuperscript{3} Zhejiang Pump Works Co. Ltd., Linhai, Zhejiang, 317000, China
\textsuperscript{4} KSB Shanghai Pump Co. Ltd., Minhang, Shanghai, 200245, China

E-mail: luoxw@tsinghua.edu.cn

Abstract. Cavitation in a condensate pump with specific speed of 95 m\textsuperscript{3}\textsuperscript{-1}s\textsuperscript{-1}min\textsuperscript{-1} was treated in this study. Cavitation performance for the pump was tested experimentally, and the steady state cavitating flows in the pump impeller were simulated by RANS method as well as a homogeneous cavitation model. It is noted that cavitating flow simulation reasonably depicted cavitation development in the pump. Compared to the tested results, the numerical simulation basically predicted later performance drops due to cavitation. Unfortunately, the cavitation simulation at the operation condition of 50\% best efficiency point could not predict the head drop up to 3\%. By applying the concept of relative cavity length, cavitation performance evaluation is achieved. For better application, future study is necessary to establish the relation between relative cavity length and performance drop.

1. Introduction

Though many useful applications of cavitation, such as ultrasound induced cavitation for drug delivery and gene therapy, etc. [1] are reported, cavitation is an unfavourable phenomenon for hydraulic machines because it usually induces performance drop, vibration, noise and so forth [2]. Due to the recent tendency that the reliability for pumps and turbines is emphasized, unsteady cavitating flows are intensively studied [3, 4, 5]. In this sense, cavitation should be avoided as possible during the industrial applications for those hydraulic machines such as the pump, the turbine, the propeller and its systems. Therefore, cavitation evaluation during design process is necessary.

Performance evaluation due to cavitation in hydraulic machines can be conducted experimentally and numerically. Since experimental evaluation is not so easy, numerical simulation is more useful at the step of product development. Thanks to the improvement of computer technology, people can treat the cavitating flow and evaluate the performance of a hydraulic machine in a short period. Though the quantitative accuracy for cavitation prediction is still limited, it is ensured that numerical simulation can predict the cavitation development fairly well, and can afford helpful hints for design optimizations by many case studies [6, 7].

In this paper, the cavitation inside a condensate pump is treated. The hydraulic performance and cavitation performance at rated rotational speed are tested experimentally. To make clear the cavitation development in the pump, the cavitating flows at several flow discharges are numerically
simulated. Based on those results, a method to evaluate performance drop due to cavitation development is proposed.

2. Test pump and its hydraulic performance
A condensate pump is designed to operate at the rated rotational speed of 1480 min\(^{-1}\). Figure 1 shows the meridional section of the pump impeller, where the diameter and width at the exit are 380mm and 25mm, and the inlet diameter is 234mm. The specific speed of the pump, i.e. \( n_s \) whose definition is shown as equation (1) at design point is 95.

\[
n_s = 3.65 \frac{nQ^{0.5}}{H^{0.75}}
\]

where \( n \): rated rotational speed; \( Q, H \): flow discharge and pump head at the rated rotational speed.

![Figure 1. Meridional section for test pump impeller.](image)

For convenience, three dimensionless parameters, i.e. flow discharge coefficient \( \phi \), pump head coefficient \( \psi \), and input power coefficient \( \tau_{in} \) are defined to express the pump performance:

\[
\phi = \frac{Q}{\pi D_2 b_2 u_2}
\]

where \( D_2, b_2 \): diameter and width at impeller exit; \( u_2 \): peripheral speed at impeller exit.

\[
\psi = \frac{H}{u_2^2 \sqrt{2g}}
\]

where \( g \): gravitational acceleration.

\[
\tau_{in} = \frac{P_{in}}{\rho \pi D_2 b_2 u_2^3 / 2}
\]

where \( \rho \): fluid density; \( P_{in} \): power input to pump through the shaft.

For the test pump, the design operation point is: \( \phi_d = 0.113, \psi_d = 0.90 \) at \( n=1480 \text{ min}^{-1} \). Figure 2 shows the experimentally tested characteristic curves. It is noted that the best efficiency point locates at \( \phi_{bep} = 0.150 \), while the corresponding head coefficient \( \psi_{bep} \) equals 0.82. There is a discrepancy in flow discharge between the design operation point \( \phi_d \) and best efficiency point \( \phi_{bep} \). The reason is that the pump is designed with a larger diameter at impeller inlet than the usual value for the sake of better cavitation performance. Thus, a wide range of flow discharge having high efficiency is achieved for the test pump.
3. Numerical methods

The numerical methods for steady state cavitating flow simulation inside the test pump are described as follows:

3.1. Governing equations

The cavitating flow is considered as the flow of a virtually homogeneous fluid including water and vapour at a certain temperature and pressure. Thus, the flow is basically controlled by mass and momentum conservation equations as equations (5) and (6).

\[
\frac{\partial \rho}{\partial t} + \frac{\partial (\rho u_j)}{\partial x_j} = 0
\]  
(5)

\[
\frac{\partial (\rho u_j)}{\partial t} + \frac{\partial (\rho u_j u_i)}{\partial x_i} = -\frac{\partial p}{\partial x_j} + \frac{\partial}{\partial x_j} \left( \mu \frac{\partial u_j}{\partial x_j} \right)
\]  
(6)

where \( u \), \( \mu \): velocity and dynamic viscosity for the virtually homogeneous fluid; \( p \): pressure.

Note that the density and dynamic viscosity for the virtually homogeneous fluid are calculated based on their volume fraction in the liquid-vapour mixture:

\[
\rho = \alpha_v \rho_v + (1-\alpha_v) \rho_l
\]  
(7)

\[
\mu = \alpha_v \mu_v + (1-\alpha_v) \mu_l
\]  
(8)

where \( v \), \( l \): subscript for vapour and liquid.

The volume fraction of vapour in the virtually homogeneous fluid is obtained by solving the following mass transfer equation.

\[
\frac{\partial (\rho \alpha_v)}{\partial t} + \frac{\partial (\rho \alpha_v u_j)}{\partial x_j} = \dot{m}^+ - \dot{m}^-
\]  
(9)

where \( \dot{m}^+ \), \( \dot{m}^- \): resource term based on bubble dynamics, and its expressions are listed in many books and references, such as Ref [8].

3.2. Computation domain and grids

Figure 3 shows the computation domain of cavitating flow simulation for the test pump. Based on the numerical tests [9], single flow passage including one impeller blade is chosen. The domain is divided into three parts: suction pipe zone upstream of the pump impeller, impeller zone, and extended passage zone downstream of the impeller. A rotating coordinate system is setting for the impeller zone with the rated rotational speed, while other parts are at a stationary coordinate system. There are interfaces among those flow parts.

Figure 2. Characteristic curves for the test pump \((n=1480 \text{ min}^{-1})\)
For better numerical accuracy, structured hexahedral meshes are generated. Table 1 shows three meshes for separated zones. Note that the mesh around the impeller blade is refined to satisfy the requirement for turbulent modelling. The distribution of $Y_{\text{plus}}$ on the blade surface is also shown in Figure 3.

![Figure 3. Computation domain and mesh generation.](image)

**Table 1.** Meshes generated for computation domain.

| Zone           | Node number | Element number |
|----------------|-------------|----------------|
| Suction pipe   | 73,800      | 68,440         |
| Impeller       | 209,670     | 194,938        |
| Extended pipe  | 108,000     | 100,949        |

3.3. Boundary and operation conditions

For cavitation simulation, total pressure i.e. $p_t$ is set at inlet plane of the computation domain. The cavitation condition, which is usually defined as $NPSH$ can be decided by equation (10).

$$NPSH = p_t - p_v$$  \hspace{1cm} (10)

At the outlet, mass flow discharge is given. For all solid walls, non-slip condition as well as wall function is applied.

The cavitation simulation is conducted at several flow discharges at the rated rotational speed. The temperature of the fluid is set the same as the experimental test.

For convenience, a commercial computational fluid dynamic code CFX is used for the calculation.

4. Results and considerations

4.1. Cavitation performance

The head drops of the test pump at two operation conditions at $n=1480 \text{ min}^{-1}$ are shown as Figure 4, where the horizontal axis is Thoma’s cavitation number defined as equation (11), and the vertical axis is head coefficient. Note that the pump head coefficient for experimental results and the impeller head coefficient (from the inlet of suction pipe to the exit of extended passage) for numerical results are used in the figure. Thus, it is reasonable that there is a difference between two results.

$$\sigma = \frac{NPSH}{H}$$  \hspace{1cm} (11)
As for the cavitation performance of the pump, cavitation specific speed i.e. $S$, whose definition is equation (12) is applied.

$$S = 3.65 \frac{nQ^{0.5}}{NPSH_{cr}^{0.75}}$$  \hspace{1cm} (12)

Note that $NPSH_{cr}$ in equation (12) is the critical $NPSH$, where the pump head drops 3% from its original value free of cavitation at the best efficiency point. Based on the experimental results, the value of $S$ is 1105.

Based on the experimental results, a gentle drop at $0.5Q_{bep}$ and a steep drop at $1.0Q_{bep}$ are obtained for the pump head coefficient curve when cavitation occurs. Compared to the experimental data, the numerical simulations predict a later cavitation development for both cases. As can be seen, no impeller head drop at $0.5Q_{bep}$ is predicted even at very low cavitation number such as $\sigma = 0.0276$.

At the case of $1.0Q_{bep}$, cavitation development predicted by numerical method is very similar to the experimental result, and the critical cavitation numbers where the head drop happens are also nearby. Unfortunately, for the case of $0.5Q_{bep}$, the impeller head drop couldn’t be predicted numerically because no convergent result of the calculation at cavitation number lower than 0.0276 was available.

### 4.2. Cavitating flow in pump impeller

To observe the cavitation development in the pump impeller, Figure 5 and Figure 6 show the cavities at the flow discharges of $0.5Q_{bep}$ and $Q_{bep}$. For each flow discharge, cavitation inception occurs at the leading edge near the suction surface of the impeller blade, and the cavity grows along the suction surface. Comparatively, at the same cavitation number, the cavity at $0.5Q_{bep}$ is smaller. For the case of $1.0Q_{bep}$ at $\sigma = 0.036$, the cavity along the suction surface extends much long in the main blade-to-blade flow passage, and a cavity close to the blade pressure surface develops from the leading edge, as shown at Figure 6(c). Consequently, the stream strikes in the flow passage is disturbed as shown in Figure 7, and the performance drop will occur.

![Cavities inside impeller](image_url)  \hspace{1cm} (a) $\sigma = 0.276$ \hspace{1cm} (b) $\sigma = 0.055$ \hspace{1cm} (c) $\sigma = 0.041$  

Figure 5. Cavities inside impeller at $0.5Q_{bep}$ for three operation conditions ($n=1480$ min$^{-1}$).
In order to correlate the performance drop with cavity size, a relative cavitation length i.e. $l_c$ is defined as the following:

$$l_c = \frac{V_{cav}}{\pi D_1 b_1 (D_2 - D_1)/2}$$

where $D_1$ and $b_1$ are diameter and flow passage width at impeller inlet. $V_{cav}$: cavity volume calculated by the integral of the cavity having vapour volume fraction of 0.1 in the pump impeller.

![Figure 6. Cavities inside impeller at 1.0$Q_{bep}$ for three operation conditions ($n=1480$ min$^{-1}$).](image)

![Figure 7. Streak lines inside impeller at 1.0$Q_{bep}$ for two operation conditions ($n=1480$ min$^{-1}$).](image)

![Figure 8. Relative cavitation length variation at two flow discharges](image)
The tendency which $l_c$ curve shows is that the relative cavity length increases rapidly with cavitation number. There seems an acceleration of $l_c$ at cavitation number lower than a turning point, such as $\sigma = 0.4$ for $1.0Q_{bep}$, and $\sigma = 0.2$ for $0.5Q_{bep}$.

It is noted that at the critical point, i.e. $\sigma_{cr} = 0.0594$ for $1.0Q_{bep}$, $l_c$ has the value of 0.035. Though performance breakdown is decided by many factors including cavity size and distribution of the cavity, it is also important to establish any relation between the cavity size and performance drop during the design stage for hydro machines. In this study, the head drop up to 3% is predicted numerically at $1.0Q_{bep}$, but is not achieved at $0.5Q_{bep}$. By using the relative cavity length, cavitation development can also be roughly featured.

There is a strange peak after cavitation number less than 0.2 for the curve of $0.5Q_{bep}$. The causes for this peak need to be investigated in future work.

5. Concluding remarks
Cavitation performance for a condensate pump was treated experimentally and numerically. Compared to the tested results, cavitating flow simulation reasonably depicted cavitation development in the pump, and predicted later performance drops due to cavitation. By applying the concept of relative cavity length, cavitation performance evaluation is also available. Future study is necessary to establish the relation between relative cavity length and performance drop.

Acknowledgments
This work was financially supported by the National Natural Science Foundation of China (Project Nos 51179091, 51206087), the Major National Scientific Instrument and Equipment Development Project (Project No. 2011YQ07004901), Tsinghua University (Grant No. 20111081106), and the China Postdoctoral Science Foundation (Project No. 2011M500314).

References
[1] Newman CMH and Bettinger T 2007 Gene Therapy 14 465
[2] Luo X 2012 Design and Optimization for Fluid Machinery (Beijing: Tsinghua University Press) p 24
[3] Brennen C E 1994 Hydrodynamics of pumps (Tokyo: Concepts ETI, Inc. and Oxford University Press)
[4] Agostino L and Salvetti M V 2007 Fluid Dynamics of Cavitation and Cavitating Turbopumps (New York: Springer)
[5] Iliescu M S, Ciocan G D and Avellan F 2008 ASME J. Fluids Eng. 130 021105
[6] Luo X, Liu S, Zhang Y and Xu H 2008 Front. Energy Power Eng. China 2(1) 31
[7] Luo X, Zhang Y, Peng J, Xu H and Yu W 2008 J. Mech. Sci. Technol. 22 1971
[8] Luo X, Ji B, Peng X, Xu H and Nishi M 2012 ASME J. Fluids Eng. 134 041202
[9] Luo X, Wei W, Ji B, Pan Z, Zhou W and Xu H 2013 J. Mech. Sci. Technol. 27 1643