Investigation of cavitation instabilities in a centrifugal pump based on one-element theory

G S Zhao\textsuperscript{1,2*} and W G Zhao\textsuperscript{1,2}

\textsuperscript{1}College of Power and Energy Engineering, Lanzhou University of Technology, Lanzhou 730050, China;
\textsuperscript{2} Key Laboratory of Fluid Machinery and System, Gansu Province. Lanzhou 730050, China.

E-mail: zhaogslut@163.com

Abstract: In order to investigate the cavitation instabilities in a centrifugal pump, the unsteady cavitating flow in a low-specific speed centrifugal pump was simulated by using a modified SST $k$-$\omega$ turbulence model combined with Kubota cavitation model. The intrinsic and system instabilities of cavitating flow were comprehensively analyzed and interpreted. For the cavity patterns formation process in a centrifugal pump, the separated flow is dominant instead of re-entrant jet flow. The transient incidence angle, cavitation number, cavitation compliance and mass flow gain factor of the middle streamline were addressed in the centrifugal pump at design point based on classical one-element theory in order to characterize the system instability. The results show that the bubbles are static stability at cavitation inception stage, which can reduce pressure fluctuations by varied cavity volume and hard to develop to rotating cavitation. It performs static instability at the stage of development because of the positive-feedback between cavity volume and flow rates, the decreasing of bubble volume would lead to the increasing of flow rate, which can induce undesirable cavitation instabilities.

1. Introduction

The leading edge cavitation is the main cavitation type in a centrifugal pump and is known to be the source of head drop. The predominate cavitation forms in a centrifugal pump represent rotating cavitation, cavitation surge, choked cavitation and other high-frequency flow instabilities. In the process from cavitation inception to fully developed cavitation, the accumulation of bubbles progressively influences the normal energy conversion. WANG et al.\textsuperscript{[1]} studied the effects of inlet incidence angle on cavitation performance based on the standard $k$-$\varepsilon$ turbulence model and visualized the leading edge cavitation patterns by experimental method. TSUJIMOTO et al.\textsuperscript{[2]} investigated the rotating cavitation instabilities in inducer based on two-element theory. With the development of computing technique, the numerical means has been a main tool to investigate the cavitating flows. SI et al.\textsuperscript{[3]} analyzed the cavitating flow instabilities in the casing of centrifugal pump based on the SST $k$-$\omega$ model.

The classical cavitation instabilities can be divided into two main classes to distinguish. One is the intrinsic instabilities, which originates in the cavity itself. As we all known, the re-entrant jet is mainly responsible for the shedding of cloud cavitation. Therefore, the re-entrant jet flow and associated cloud cavitation is clearly of intrinsic type. The second instabilities are the system instabilities. For system instabilities, the dynamic behavior of the cavity depends drastically upon its environment, such as the...
response to overall pressure pulsations and flow rates \(^4\).

Traditional centrifugal pumps were designed based on one-element theory. The inlet incidence angle of middle streamline is usually designed positive to improve the cavitation performance. Different modes of system cavitation instabilities have been observed and reported in a centrifugal pump. One of them occurs at the operation before head drop situation and is caused by the vapor volume fluctuation, which performs rotating cavitation and cavitation surge to cause severe results under continuous impingement even at the designed point. This work comprehensively investigates re-entrant jet intrinsic cavitation instabilities and system instabilities in a low-specific speed centrifugal pump by the numerical method combined with experiments to validate the numerical accuracy, which is helpful to control cavitating flow and the optimize design of centrifugal pumps.

2. Numerical method and cavitation model

The numerical simulation employed the modified SST \(k-\omega\) model to solve the unsteady Navier-Stokes equations coupled with Kubota cavitation model \(^5\). The continuity and the momentum equation for the mixture are given as follows:

- **Continuity equation**
  \[
  \frac{\partial \rho_m}{\partial t} + \frac{\partial (\rho_m u_i)}{\partial x_i} = 0
  \]

- **Momentum equation**
  \[
  \frac{\partial (\rho_m u_i)}{\partial t} + \frac{\partial (\rho_m u_i u_j)}{\partial x_j} = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j} \left[ \left( \mu + \mu_t \right) \frac{\partial u_i}{\partial x_j} + \frac{\partial u_i}{\partial x_i} - \frac{2}{3} \frac{\partial u_i}{\partial x_i} \delta_{ij} \right]
  \]

where \(u_i, u_j\) and \(u_k\) are the velocity component, \(\rho_m\) is the mixture density, \(\delta_{ij}\) is the Kronecker delta, \(\mu\) is the laminar viscosity, \(\mu_t\) is the turbulence viscosity, which is closed by SST \(k-\omega\) model. Here, the mixture density \(\rho_m\) is defined as:

\[
\rho_m = \alpha \rho_v + \rho_l (1-\alpha_c)
\]

where \(\alpha\) is the volume fraction of one component, the subscripts \(v\) and \(l\) refer to the component of vapor and liquid, respectively. To improve the simulation accuracy by considering the compressibility of cavitating flow, and capture the dynamic characteristics of cavity flow, some modification is necessary only for the viscosity term in equation (2) and defined density function \(f(\rho)\) to replace the mixture density \(\rho_m\). The modified term is defined as:

\[
\mu_t = f(\rho) \frac{k}{\omega}
\]

\[
f(\rho) = \rho_v + \frac{(\rho_m - \rho_v)^n}{(\rho_l - \rho_v)^n}; n >> 1
\]

where \(n\) is usually 10. The cavitation process is governed by the following mass transfer equation:

\[
\frac{\partial (\rho_{m f_i})}{\partial t} + \frac{\partial (\rho_{m u_i} f_i)}{\partial x_i} = R_v - R_c
\]

\[
R_v = C_v \frac{3 \alpha_{nu} (1-\alpha_c) \rho_l}{R_b} \left( \frac{2}{3} \frac{p_v - p}{\rho_l} \right); p < p_v,
\]

\[
R_c = C_c \frac{3 \alpha_{nu} \rho_l}{R_b} \left( \frac{2}{3} \frac{p - p_v}{\rho_l} \right); p > p_v,
\]

where \(f_i\) is the vapor mass fraction, \(R_c\) is the liquid vaporization rate, \(R_v\) is the vapor condensation rate, \(R_b\) is the simplified bubble radius, \(\alpha_{nu}\) is the volume fraction of nuclei, \(C_v\) and \(C_c\) are empirical factors for the vaporization and the condensation processes with recommended values of 50 and 0.01. \(\alpha_{nu}\) has the value of \(5 \times 10^{-4}\) and \(R_b\) is the typical bubble size with the value of \(1 \times 10^{-6}\).
3. Computational domain and boundary conditions

The research model is a low specific-speed centrifugal pump with suction pipe, impeller and volute. The specific-speed $n_s=32$ is defined at the design point ($n_s=3.65nQ^{0.5}H^{0.75}/60$, $H=4.2$ m, $n=500$ r/min, $Q=8.6$ m$^3$/h). The main geometric parameters are shown in table 1, whose blade profile is cylindrical shape.

| Suction diameter $D_1$/mm | Discharge diameter $D_0$/mm | Inlet diameter $D_1$/mm | Outlet diameter $D_2$/mm |
|--------------------------|-----------------------------|------------------------|-------------------------|
| 90                       | 70                          | 80                     | 310                     |
| Outlet width $b_2$/mm    | Inlet blade angle $\beta_1$/° | Outlet blade angle $\beta_2$/° | Blade number $Z$         |
| 12                       | 37                          | 37                     | 6                       |

Table 1. Main parameters of model pump

Structural hexahedral meshes are generated in the suction chamber, impeller domain of the centrifugal pump, and the meshes near the blade surface are locally refined to get better calculation resolution. Through mesh independency test as shown in table 2, determine the final scheme in which the total number of nodes and elements is 1112420 and 1180228, respectively. The whole computational domain and mesh are shown in figure 1. The monitoring point m is located at the blade inlet of middle streamline as shown in figure 2.

| Coarse      | Impeller     | Volute      | Total       | Head (m) |
|-------------|--------------|-------------|-------------|----------|
| Suction pipe | 104238       | 426300      | 156272      | 686810   | 4.43     |
| Medium      | 145236       | 531810      | 196168      | 873214   | 4.57     |
| Fine        | 145236       | 838824      | 196168      | 1180228  | 4.58     |

Table 2. Check of grid independency

Figure 1. Computational domain

Figure 2. Monitoring points

This calculation was completed by commercial package CFX 15.0, the modified SST $k-\omega$ was implemented by user defined function. The total pressure was specified at the inlet of the suction pipe and the volume fraction of vapor was set to zero for the cavitation case at the suction inlet. The mass flow rate boundary condition was given at outlet. Sliding mesh technology was employed to simulate the transient flow. In the steady calculation, the interface type between rotating domain and stationary domain was set as frozen rotor. The steady results are set as the initial value for the transient simulation to accelerate the convergence. Meanwhile, the interface type is reset to transient frozen rotor. The time step $\Delta t$ is set to 0.001s of every three degrees for a rotating cycle, which means this time-step is equivalent to 120 time-steps per impeller revolution. The total time of the unsteady calculation is set to 2 rotating periods when the results have been stable and reliable. The iterative maximum residual for every time step was set to $1\times10^4$. 


4. Results and discussion

4.1. Comparison of hydraulic performance

To validate the accuracy of numerical method, the experiment was conducted on the testing stand of external performance of centrifugal pump in Lanzhou University of Technology as shown in figure 3.

![Figure 3. Closed centrifugal pump test](image)

The cavitation number is employed to describe the possibility of cavitation occurrence and defined as:

$$\sigma = \frac{p_1 - p_v}{\frac{1}{2} \rho U^2}$$  \hspace{1cm} (9)

where $p_1$ is the reference pressure and the inlet pressure of pump; $p_v$ is the saturated vapor pressure; $U$ is the reference velocity, which is the circumferential velocity of intersection of blade and the shroud and defined as:

$$U = \frac{n \pi D_1}{60}$$  \hspace{1cm} (10)

where $n$ is the rotating speed of impeller; $D_1$ is the diameter of intersection of blade inlet and shroud.

![Figure 4. External performance [Exp., Num.,]](image)  \hspace{1cm} ![Figure 5. Cavitation performance [Exp., Num.,]](image)

The predicted external performance agrees fairly well with experiment counterpart as shown in figure 4. The trend of cavitation performance also agrees well with experiment observations. However, there is still some deviation between calculation results and experimental observations because the calculation neglects many factors that influence the real cavitating flow such as water quality, incompressible effects and so on. Besides, whatever in the process of calculation or experiment, the inflexion point of cavitation curve is very sensitive to boundary conditions. The cavitation number of numerical broken head is 0.09 as shown in figure 5. This investigation studied the cavitation...
instabilities before broken head, namely $\sigma=1.16$, $\sigma=0.82$, $\sigma=0.24$ and $\sigma=0.15$ operating conditions.

4.2. Intrinsic instabilities of cavitating flow

Figure 6 shows the cavitating regions and the impeller for different cavitation numbers before the head drop of the centrifugal pump. The predicted cavity shape is illustrated by the iso-surface of vapor volume fraction of 10%. The cavitation phenomenon occurs when the cavitation number equals 1.16, the smaller the cavitation number is, the larger the vapor volume is.

![Cavity patterns]

Figure 6. Cavity patterns

The cavitation performs semi-transparent sheet pattern, which occurs at the leading edge of the impeller for cavitation inception $\sigma=1.16$ and 0.82 as shown in figure 6(a)-(b). For the stage of cavitation developing $\sigma=0.82$, the cavitation performs shedding cloud pattern and asymmetrically distribute in the inter-blade passages as shown in figure 6(c), which is an adverse cavitation form that could cause severe damages in a centrifugal pump. The cloud cavitation developed further when the cavitation decreases as shown in figure 6(d), the blade inlet is blocked by the cavities, which would broke the normal process of energy exchange.

The reason for the cavities shedding is the re-entrant flow towards the leading edge of the cavitating interface, which has been solidly proved on the series conducted hydrofoil experiments by KAWANAMI [6]. It has been ensured that the jet velocity is of the same magnitude as the upstream main flow, otherwise it doesn’t work to be responsible for the shedding formation. We also detect the velocity vector distributions in the centrifugal pump for the $\sigma=0.24$, unfortunately, the re-entrant is too weak to cut off the developed cavitation as shown in figure 7(a). JI et al. [7] have revealed the cloud cavitation mechanism induced by the re-entrant jet flow towards cavity interface on the leading edge of a twisted hydrofoil and found the effective cutoff velocity of re-entrant jet is identical or very close to the main flow as shown in figure 7(b). Actually, an obstacle have ever been arranged on the suction surface to stop the re-entrant jet flow in order to prevent the cloud cavitation instabilities [6]. The authors have ever employed the same method to control cavitation in a centrifugal pump, but it seems doesn’t work to suppress the cavitation. The reason is of interest and can be explained that the re-entrant is not dominant in the centrifugal pump, so the obstacles method is invalid to some extent.
4.3. Time sequence analysis of system instabilities

The modes of cavitation system instabilities perform choked cavitation and cavitation surge etc. These instabilities occur at the various cavitation numbers before the head drop, which varied with the increase of vapor volume $V_{\text{cav}}$ induced by the incidence angle $\alpha$. There are two essential parameters introduced in one-element theory to analyze rotating cavitation instabilities, one is the cavitation compliance that describes the feedback between pressure fluctuations $p$ and the vapor volume $V_{\text{cav}}$, the other is the mass flow gain factor that describes the relationship of the incidence angle $\alpha$ and vapor volume $V_{\text{cav}}$. Some parameters utilized for the description of system instability are defined as follows:

Incidence angle $\alpha$

$$\alpha = \beta_i - \beta'_i$$

where $\beta_i$ is the blade inlet angle, $\beta'_i$ is the relative flow angle at the blade inlet.

The vapor volume $V_{\text{cav}}$

$$V_{\text{cav}} = \sum_{i=1}^{N} \alpha_{v,i} \cdot V_i$$

where $N$ is the total number of control volumes in the computational domain, $\alpha_{v,i}$ is vapor volume fraction in each control volume and $V_i$ is the volume of each cell.

The cavitation number $\sigma_m$ at blade inlet

$$\sigma_m = \frac{p_m - p_v}{\frac{1}{2} \rho U_m^2}$$

where $p_m$ is the absolute pressure at the monitoring point, $p_v$ is the saturated vapor pressure, $U_m$ is the circumferential velocity of the point $m$, and it can be expressed as:

$$U_m = \frac{n \pi D_m}{60}$$

where $n$ is the rotating speed of impeller ($r/min$); $D_m$ is the diameter of point $m$.

Cavitation compliance $K$

$$K = -\frac{dV_{\text{cav}}}{dp_m}$$

Mass flow gain factor $M$

$$M = \frac{dV_{\text{cav}}}{d\alpha}$$

According to the theory of cavitation instabilities, the cavitation compliance $K$ was employed to describe the sensitivity of the vapor volume corresponding to ambient pressure. When the $K$ value is lower than zero, the vapor volume decrease if the pressure decrease, the space the bubbles shrink is covered by the surround fluids, which lead to the pressure decrease further and deteriorate the
cavitating flow. Therefore, the cavity presents static unstable for $K<0$. The $K>0$ case means the vapor volume would decrease if the pressure increase, the pressure fluctuations excited by cavitating flow can be adjusted through the variations of vapor volume, which is a common case for cavitation process. The mass flow gain factor $M$ is employed to describe the sensitivity of the vapor volume corresponding to mass flow. If the flow space decreased by cavity thickness need to be filled by fluids, which leads to the increase of mass flow and decrease of incidence angle. The decreased incidence angle leads to the further increasing of blade inlet mass flow, which reflects $M>0$ and this positive feedback type is the source of cavitation surge and rotating cavitation [8].

Figure 8. Cavitation number $\sigma_m$

Figure 9. Incidence angle $\alpha$

Figure 8 shows time sequence of $\sigma_m$, there are 6 crests and troughs in one rotating cycle in accordance to the blade numbers. Figure 9 shows the inlet incidence angle of the middle streamline and all performs positive in one rotating cycle. Basically, the cavitation occurs at the suction side of blades for the positive incidence angle and occurs at the pressure side of blades for the negative incidence angle, respectively. It can be obviously seen that the fluctuations of incidence angle become larger with the decrease of cavitation number. Such large incidence angle is easy to cause strong adverse pressure gradient and the flow separation in the zone of blade backside.
Figure 10. Cavity volume $V_{cav}$

The vapor volume doesn’t perform periodicity in the cycle as shown in figure 10. The vapor volume behaviors alternate law in one cycle for $\sigma=1.16$ and $\sigma=0.82$. The vapor volume firstly decreases and then increases for $\sigma=0.24$. When the cavitation number is 0.15, the volumes keep attenuating. It needs to pay more attentions on the relationship between volume variations and pressure fluctuations in pumps.

Figure 11. Cavitation compliance $K$

Figure 11 shows the cavitation compliance variation in one cycle. For the cavitation inception stage $\sigma=1.16$, the $K$ is almost higher than 0 in the whole cycle, which means the expanded cavity caused by decreased ambient pressure squeeze the surround liquids to prevent the further decrease of pressure when pressure dropped. Therefore, the cavity stays at the static stable for cavitation inception. For the condition of $\sigma=0.82$, most area of histogram of $K$ value are lower than 0 in the cycle, the vapor volume decreases if the pressure decreases, which lead to the surround liquids fill the region of cavity shrink and expand the low pressure region, the cavity stays static unstable situation. However for the case of
σ=0.24, The \( K \) value alternates between \( K>0 \) and \( K<0 \), which performs unstable oscillation behaviors. When the cavity developed further (\( \sigma=0.15 \)), the \( K \) value performs huge amplitude and periodic feature. The volume variation is a main factor to maintain the stabilities of cavity. Because of the positive periodic \( K \) value, the cavity mode is typical of cavitation surge situation \([9]\).

![Figure 12. Mass flow gain factor \( M \)](image)

Figure 12 shows the variations of mass flow gain factor in one cycle. For the cavitation inception \( \sigma=1.16 \), the \( M \) value performs \( M<0 \) in almost one cycle, which the decrease of vapor volume wouldn’t cause the increase of inlet mass flow and hard to develop to rotating cavitation. For the situation of \( \sigma=0.82 \) and \( \sigma=0.24 \), the \( M \) value experiences the alternative between positive and negative, and the amplitude of \( M \) becomes larger with the decrease of cavitation number. For the case of \( \sigma=0.15 \), the positive \( M \) value is dominant, which means the decreased vapor volume lead to the decease of incidence angle corresponding to the increasing of flow rate, which means there is positive feedback between cavity volume and flow rate and can be regarded as the source of rotating cavitation \([10]\).

4.4. Analysis with the one-dimensional model

A simplified one-dimensional model is often employed to analyze system instabilities \([11]\). Then, the continuity equation between the inlet and outlet flow rates of this pump (\( Q_1 \) and \( Q_2 \), respectively) is:

\[
Q_2 - Q_1 = \frac{dV_{cav}}{dt}
\]  \hspace{1cm} (17)

The relation between flow rates difference and the first derivative of the cavity volume are plotted in figure 13.
Figure 13. Comparison of flow rates difference and volume growth rate

As indicated in figure 13, the flow rate difference matches very well with the first derivative of cavity volume for $\sigma=1.16, 0.82, 0.24$, respectively. For $\sigma=0.15$, although there are some deviations, the tendency of the two variables are similar. According to the above discussion of flow structure, the cavity performs shedding characteristics and transient forms from 2D to 3D in the $\sigma=0.15$ period, which would be beyond the assumption of one-dimensional theory to some extent [13]. The three-dimensional cavity performs strong instabilities and brings about severe damages to the material, even causes undesirable vibration and noise for the machine, which are urgently asked to avoid in various engineering occasions.

5. Conclusion
The inlet cavitation number $\sigma_m$ performs wave situation in one cycle, the smaller $\sigma_m$ is, the lager the fluctuation is. The amplitude of $K$ and $M$ becomes larger with the decrease of cavitation number, which means the cavitation instabilities become severe corresponding to cavity volume. For the stage of head affected by cavity volume, the cavitating flow performs strong three-dimensional flow character.

The intrinsic instabilities of re-entrant jet associated cloud cavitation are not clear in centrifugal pumps. The magnitude of jet velocity is very small compared to the main flow in a passage and can’t form effective intensity to cut off the developed cavity.

As to the system instability, the positive cavitation compliance means the vapor volume statically stable according to classical one-element theory. At the instant of cavitation inception, the bubbles are statically stable because of $K>0$, which means it can decrease the pressure fluctuation by vapor volume variation. For the process of cavitation developing associated to $K<0$, the bubbles stay unstable situation, which means the accumulation of bubbles make pressure decrease further to deteriorate the flow structure.
For the cavitation inception related to $M<0$, the mass flow become low if the vapor volume increase, which means it is hard to induce rotating cavitation. On the contrary, the alternate between $M>0$ and $M<0$ is corresponding to the cavitation developing, the positive feedback exists between vapor volume and mass flow rate, which means the enlarged vapor volume induces the decrease of inlet mass flow to cause severe rotating cavitation. The combined effects of $K$ and $M$ on system cavitation instabilities in centrifugal pump need to be taken the equilibriums between $K$ and $M$ into account.

Acknowledgments
This work was financially supported by the National Natural Science Foundation of China, (Project No. 51269011)

References
[1] Wang Y., Liu H.L., Wang J., et al. 2013, Experimental measurement on cavitation pattern at impeller inlet of centrifugal pump [J]. Transactions of the Chinese Society of Agricultural Machinery, 07: 45-49.
[2] Tsujimoto Y, Kamiyo K, Yoshida Y. 1993, A theoretical analysis of rotating cavitation in inducers [J]. ASME Journal of Fluid Engineering, 115:135-141.
[3] Si Q.R., Yuan S.Q., Li X.J., et al. 2014, Numerical simulation of unsteady cavitation flow in the casing of a centrifugal pump [J]. Transactions of the Chinese Society of Agricultural Machinery, 05: 84-91.
[4] Franc J.P. 2001, Partial cavity instabilities and re-entrant jet [C]. International Symposium on Cavitation.
[5] Zwart P J, Gerber A G, Belamri T. 2004, A two-phase flow model for predicting cavitation dynamics [C]. Proceedings of International Conference on Multiphase Flow, Yokohama, Japan.
[6] Kawanami Y, Kato H, Yamaguchi M, et al. 1997, Mechanism and control of cloud cavitation [J]. ASME Journal of Fluids Engineering, 119(4): 788-794.
[7] Ji B., Luo X.W., Wu Y.L., et al, 2013, Numerical analysis of unsteady cavitating turbulent flow and shedding horse-shoe vortex structure around a twisted hydrofoil [J]. International Journal of Multiphase Flow, 51 (5) :33-43
[8] Pan Z.Y., Yuan S.Q. 2013, Fundamentals of cavitation in pumps [M], Jiangsu University Press.
[9] Franc J.P., Michel J.M. 2005, Fundamentals of cavitation [M]. Springer Press.
[10] Tsujimoto Y., Kamiyo K. and Brennen C.E. 2001, Unified treatment of cavitation instabilities of turbomachines [J]. Journal of Propulsion and Power, 17(3): 636-643.
[11] Chen C.K., Nicolet C, Yonezawa K, et al. 2007, One dimensional analysis of full load draft tube surge [J], ASME Journal of Fluids Engineering, 130 (4):391-396.
[12] Ji B., Luo, X.W., Ardnt R.E., et al. 2015. Large eddy simulation and theoretical investigations of the transient cavitating vertical flow structure around a NACA66 hydrofoil [J]. International Journal of Multiphase Flow, 68, 121-134.