

Pressure fluctuation characteristics induced by cavitation in a centrifugal pump

W T WANG¹, H LU² and G Q Q G MENG^{1*}

¹ Inner Mongolia Vocational College of Chemical Engineering, Hohhot 010070, Inner Mongolia, China

² Baotou Light Industry Vocational Technical College, Baotou 014035, Inner Mongolia, China

E-mail: erenmangha2012@163.com

Abstract. The unsteady cavitation flows in a centrifugal pump are numerically simulated by using the RNG k - ε turbulence model and the improved mass transport cavitation model. The calculated data is in good agreement with the experimental results, so the numerical modal and method can accurately predict the cavitation flows in a centrifugal pump. The flow characteristics without and with cavitation are analyzed by setting 15 monitoring points in the impeller. The calculation results indicated that the dominant frequencies of pressure fluctuations in impeller is the impeller rotating frequency f_i and $2f_i$, the impeller rotating frequency always played the dominant role. Under non-cavitation condition, the maximum value of pressure fluctuation amplitudes appears at impeller outlet. Under cavitation condition, the maximum value of pressure fluctuation amplitudes appears near 4/5 blade length from blade inlet. The shedding and collapsing of cavitation bubbles in impeller, which disturbs the flow field of pump impeller and causes the amplitude of pressure fluctuations increase. This research provided a reference for optimization design of impeller in centrifugal pump.

1. Introduction

Cavitation is a common phenomenon in the hydraulic machinery operating and one of the important causes of pressure fluctuation. Pressure fluctuation not only produces noise and vibration, but also causes the flow component alternating load, causing its fatigue damage. At present, the study of hydraulic mechanical cavitation characteristics is mainly carried out from the experimental research [1-3] and numerical simulation [4-6]. Cernetic [7] adopts acceleration sensor and microphone, and the initial cavitation of centrifugal pump is monitored. Duplaa et al. [8] tested a cavitating centrifugal pump during fast startups, and large-scale low frequency vibration pressure signal is captured under the large flow rate, and the water hammer phenomena is observed under the low flow rate. Ding et al. [9] calculated the pump cavitation flow field using the standard k - ε turbulence model and the full cavitation model, and considered that the relationship between gas content and the pressure, the fluid compressibility effects and non-uniformity of noncondensable gas mass fraction. The calculation result indicates predicted pump performance and cavitation characteristics match well with the test results. Shi et al. [10] numerical simulated the unsteady flow of centrifugal pump, the results show that the cyclic and static coupling between the impeller and volute is the main factor causing the pressure fluctuation, and the amplitude of pressure fluctuations increase with the development of cavitation. Liu



et al. [11] applied FBM model and modified mass transport cavitation model, and successfully predicted the cavitation bubbles patterns of centrifugal pump impeller inlet.

In this paper, the unsteady cavitation flow in a centrifugal pump are calculated by using the RNG k - ε turbulence model and improved mass transport cavitation model, and verify the calculation results through test. The pressure fluctuation characteristics is analyzed, the evolution process of cavitation patterns in the impeller is also discussed.

2. Computational domain and grid

2.1. Computational model and method

The continuity and momentum equations are as follows:

$$\frac{\partial}{\partial t}(\rho_m) + \nabla(\rho_m U) = 0 \quad (1)$$

$$\frac{\partial}{\partial t}(\rho_m U) + \nabla(\rho_m UU) = -\nabla p + \nabla[(\mu_m + \mu_t)\nabla U] + \frac{1}{3}\nabla[(\mu_m + \mu_t)\nabla U] \quad (2)$$

Where U is the velocity vector, p is the pressure, μ_t is the turbulent viscosity, ρ_m is the mixture density and μ_m is the mixture dynamic viscosity.

The turbulence model adopts RNG k - ε equation.

In the present study, the mass transport cavitation model is selected.

$$\frac{\partial}{\partial t}(\alpha_v \rho_v) + \nabla(\alpha_v \rho_v U) = m^+ - m^- \quad (3)$$

Where α_v is the vapor volume fraction, ρ_v is the vapor density, which value is 0.554 kg/m^3 , m^+ is the source term for evaporation, and m^- is the source term for condensation.

The calculation formula of m^+ and m^- are shown in Eq. (4) and (5).

$$m^+ = C_{\text{evap}} \frac{3\alpha_v \rho_v}{R_b} \sqrt{\frac{2}{3} \frac{\max(p_v - p, 0)}{\rho_l}} \quad (4)$$

$$m^- = C_{\text{cond}} \frac{3\alpha_v \rho_v}{R_b} \sqrt{\frac{2}{3} \frac{\max(p - p_v, 0)}{\rho_l}} \quad (5)$$

Where ρ_l is the liquid density, R_b stands for the nucleation site radius, p_v is the water vaporization pressure. The values of these parameters are: $\rho_l=997 \text{ kg/m}^3$, $R_b=10^{-6} \text{ m}$, $p_v=3574 \text{ Pa}$. C_{evap} and C_{cond} are the empirical coefficients of evaporation and condensation, which values are 50 and 0.01 [12], respectively. The value of the empirical coefficient of condensation is modified, and adopted $C_{\text{cond}}=0.0001$ [13] to improve the quality of the transport cavitation model.

The Ansys-CFX14.0 commercial software is used for numerical simulation, and the pump inlet and outlet are given the total pressure and mass flow rate, respectively. The interface between the stationary and rotating frame is set to the transient frozen rotor mode. In steady calculation, the inlet pressure value is given by the test results and gradually decreases, and the current calculation point result is the initial value of the next calculation point. In unsteady calculation, one impeller rotation cycle time is $T=0.04138 \text{ s}$, using steady calculated results as the initial value, and based on the number of blades $Z_i=7$, determine the time step is $\Delta t=T/7/32=0.0001847 \text{ s}$.

2.2. Parameter and grid of centrifugal pump

The centrifugal pump used in this paper is a pump with a shrouded impeller and seven twisted blades. The main parameters are given as follows: the volume flow rate $Q_d=25 \text{ m}^3/\text{h}$, the head $H=7 \text{ m}$, the rotational speed $n=1450 \text{ r/min}$, the number of blades $Z_i=7$, the impeller outer diameter $D_2=160 \text{ mm}$. The computational domain consists of inlet section, impeller area and volute. The whole computational domain grid of the centrifugal pump adopts structured hexahedral grid, the impeller grid, as shown in figure 1. The grid independence has been verified, the grid with 1.56 million elements is employed for the following calculations [13].

In order to analysis the pressure fluctuation characteristics of pump impeller, 15 monitoring points are set in the impeller inside. The monitoring points positions are as follows: the blade suction side (S1, S2, S3, S4, S5), the flow passage middle (M1, M2, M3, M4, M5) and the blade pressure side (P1, P2, P3, P4, P5), as shown in figure 2.



Figure 1. Centrifugal pump impeller grid.

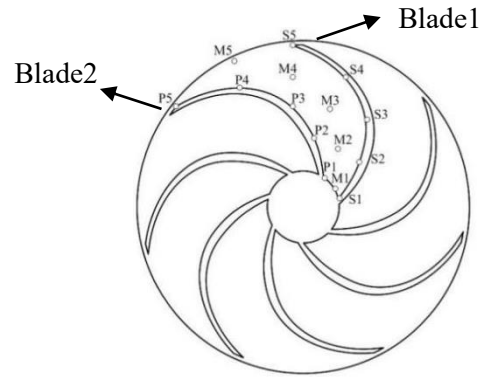
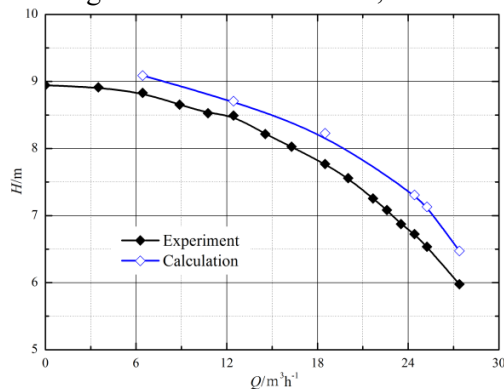


Figure 2. Monitoring points position in impeller.

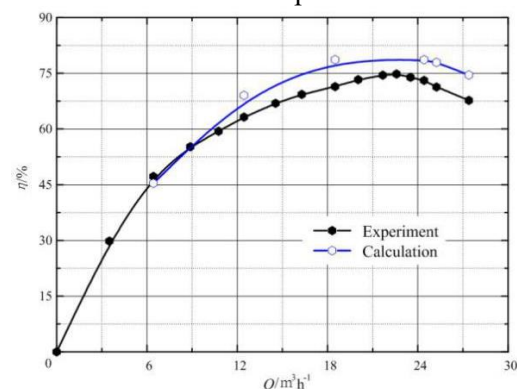
3. Results and analysis

3.1. Validation of calculation results

The pump performance numerical results is compared with the experimental data, as shown in figure 3. In the range of 6.434–27.393 m³/h, the calculated results is consistent with the experimental results.



(a) Centrifugal pump head



(b) Centrifugal pump efficiency

Figure 3. Centrifugal pump head and efficiency.

In the unsteady calculation condition, the numerical results of pump cavitation performance is compared with the experimental data, as shown in figure 4. The vertical axis is the pump hydraulic head, the horizontal coordinates is the available net positive suction head (*NPSHa*) which is defined in equation (6).

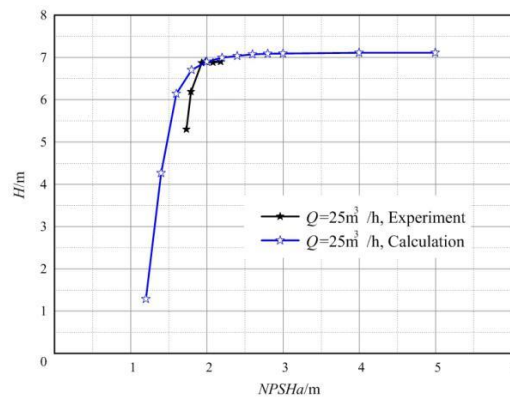


Figure 4. Centrifugal pump head and $NPSHa$.

$$NPSHa = \frac{p_{in}}{\rho_l g} + \frac{u_{in}^2}{2g} - \frac{p_v}{\rho_l g} \quad (6)$$

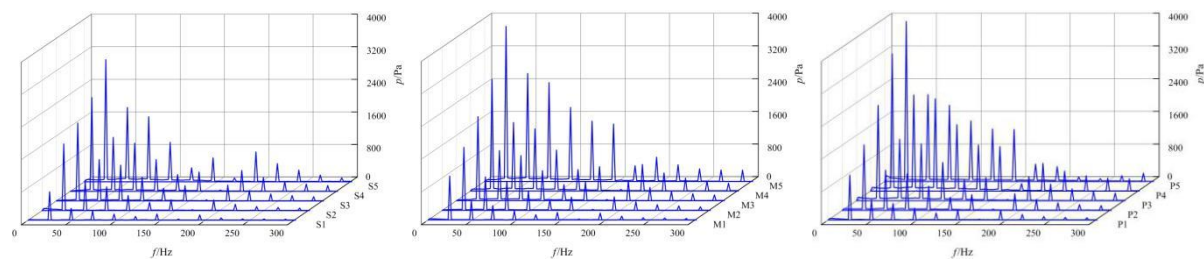
Where p_{in} is the pump inlet pressure and u_{in} is the pump inlet speed.

The result showed the change curve of pump head which obtained by numerical calculation is basically consistent with the experimental results.

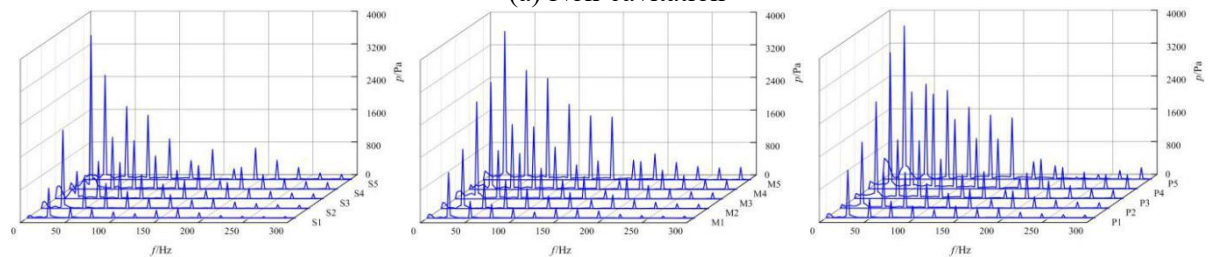
3.2. Analysis of the pressure fluctuation frequency domain in impeller

In the calculation of unsteady cavitation flows, the full developed condition $NPSHa=1.80$ m and $NPSHa=1.40$ m are selected to analysis. The total time of calculation is 10 impeller rotation cycles. The characteristics of pressure fluctuation frequency domain is obtained by FFT (Fast Fourier Transform). The impeller rotating frequency is $f_i=24.17$ Hz.

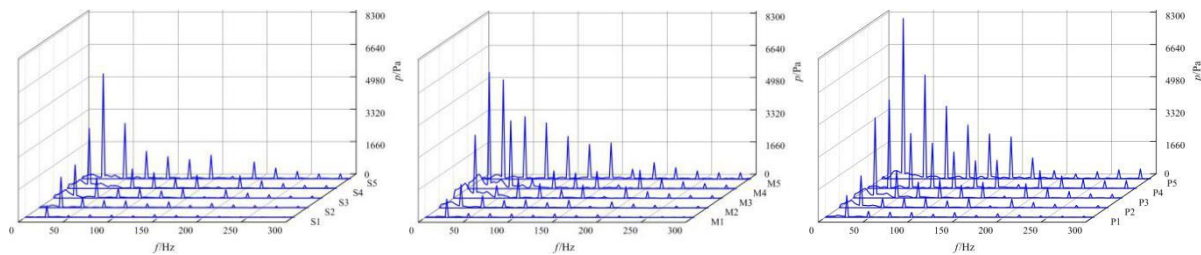
Figure 5 shows the frequency characteristics of pressure fluctuations under non-cavitation and cavitation condition. Under non-cavitation, the pressure fluctuation dominant frequency on each monitoring point is impeller rotating frequency f_i ; Under $NPSHa=1.80$ m, the dominant frequency on monitoring point S3 is $2f_i$ and the other points is f_i ; Under $NPSHa=1.40$ m, the dominant frequency on each monitoring point is impeller rotating frequency f_i . Under non-cavitation and cavitation condition, the amplitudes of pressure fluctuation on each monitoring point decreases with the frequency increase.



(a) Non-cavitation



(b) $NPSHa=1.80$ m

(c) $NPSHa=1.40$ m**Figure 5.** Pressure fluctuation frequency domain in impeller.

3.3. Analysis of the maximum amplitude of pressure in impeller

Table 1 shows the maximum amplitudes of pressure fluctuations for the non-cavitation and cavitation condition. Under non-cavitation, the pressure fluctuation maximum amplitudes on each monitoring point in the impeller is increasing from the inlet to outlet, which is the biggest at the impeller outlet. Under $NPSHa=1.8$ m, the pressure fluctuation maximum amplitudes in blade suction side is the biggest on the monitoring point S4, which value is 3756.1 Pa, about 1.6 times of non-cavitation. It is in the flow passage and blade pressure side, the biggest on the monitoring point M5 and P5, which values are 3643.3 Pa and 3736.5 Pa respectively, and basically consistent with non-cavitation. Under $NPSHa=1.4$ m, the pressure fluctuation maximum amplitudes in blade suction side is the biggest on the monitoring point S5, which value is 5388.8 Pa, about 1.8 times of non-cavitation. It is in the flow passage and blade pressure side, the biggest on the monitoring point M4 and P5, which values are 5997.8 Pa and 8223.8 Pa respectively, and about 2.2 and 2.1 times of non-cavitation.

Table 1. Maximum amplitude of pressure in impeller.

Monitoring points	Maximum amplitude of pressure fluctuation/Pa		
	Non-cavitation	$NPSHa=1.80$ m	$NPSHa=1.40$ m
S1	691.9	731.8	584.6
S2	1 632.8	1 909.4	1 569
S3	1 914.1	667.2	1 693.2
S4	2 301.6	3 756.1	3 075.4
S5	3 002.6	2 551.2	5 388.8
M1	1 071.9	1 119.6	942.5
M2	1 542.6	1 457.8	1 201.9
M3	2 058.2	2 386	3 250.9
M4	2 726.8	2 626.8	5 997.8
M5	3 793.4	3 643.3	5 107.3
P1	1 096.2	1 161.8	1 122.5
P2	1 603.2	1 606.4	1 632.7
P3	2 331.4	2 358	4 107.7
P4	3 359.6	3 320.9	4 541.6
P5	3 913.9	3 736.5	8 223.8

3.4. Analysis of the cavitation patterns in impeller

Table 1 shows that compared with non-cavitation, the amplitude of pressure fluctuations has changed greatly in the cavitation condition. Under $NPSHa=1.8$ m, the maximum amplitudes in blade suction side is the biggest at the monitoring point S4, not at the outlet of impeller. Under $NPSHa=1.4$ m, the maximum amplitudes in the flow passage is the biggest at the monitoring point M4, not at the impeller outlet.

The pressure distribution on the suction and pressure side of the blades under $NPSHa=1.8$ m and 1.4 m, as shown in figure 6 and 7.

Under $NPSHa=1.8$ m and 1.4 m, instantaneous pressure distribution on the suction side of the blade1, as shown in figure 6. The pressure distribution on the front of blade suction side is uneven and on the rear of blade suction side is more uniform. With time, the pressure distribution on the front of blade suction side has changed greatly, especially near the top of suction side.

Under $NPSHa=1.8$ m and 1.4 m, instantaneous pressure distribution on the pressure side of the blade2, as shown in figure 7. The pressure distribution on the pressure side is more uniform except for near the inlet. With time, the pressure distribution on the pressure side is relatively stable.

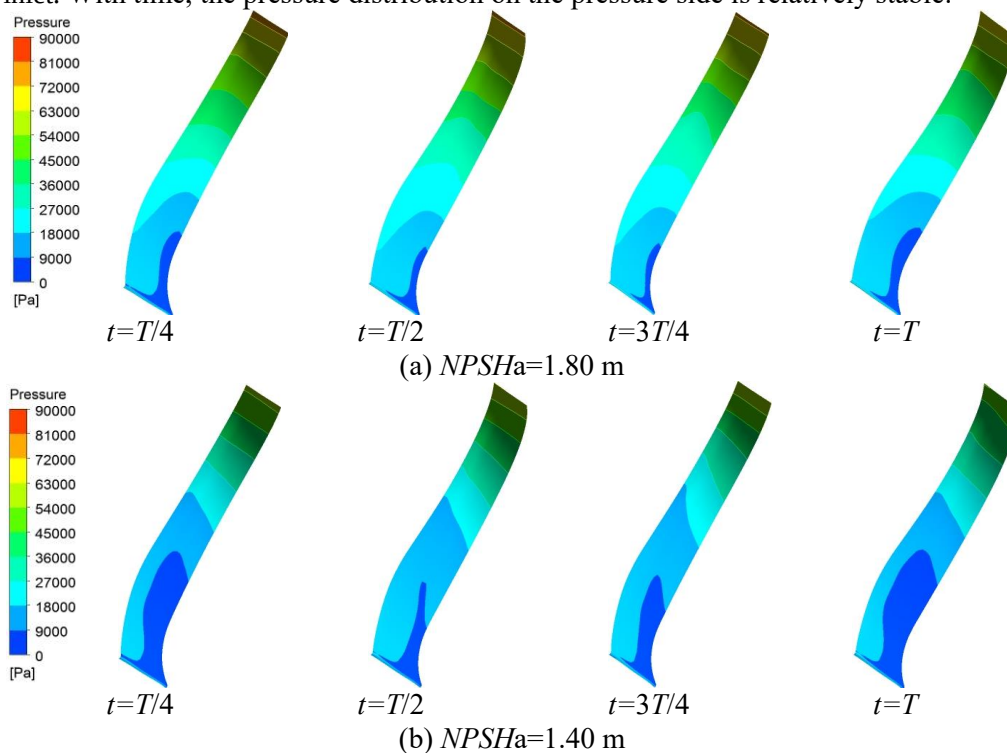
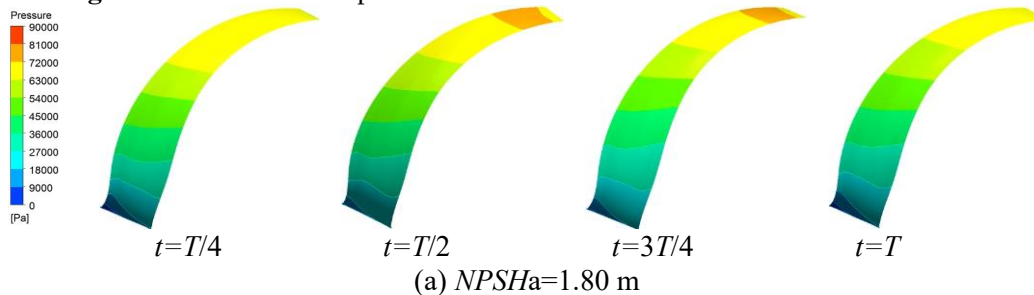


Figure 6. Instantaneous pressure distribution on the suction side of the blade1.



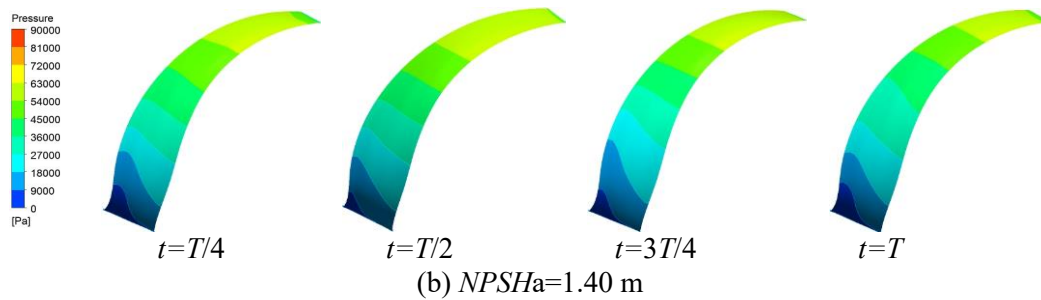


Figure 7. Instantaneous pressure distribution on the pressure side of the blade2.

The cavitation transient patterns in the impeller flow passage under $NPSHa=1.8$ m and 1.4 m, as shown in figure 8. The cavitation patterns in the No.1 impeller flow passage (Corresponding the orange blade1 suction side) is analyzed. The cavity volume fraction is 0.1.

Under $NPSHa=1.8$ m, the evolution process of cavitation patterns, as shown in figure 8(a). When $t=T/4$, a continuous cavitation bubbles attached to the blade suction side of the No.1 flow passage, the trailing edge of cavitation bubbles extends to the monitoring point S3; When the No.1 flow passage rotates along the direction of impeller rotation to $t=T/2$, $3T/4$, the cavitation bubbles growths along the blade suction side towards the impeller outlet; When $t=T$, the trailing edge of cavitation bubbles extends to the monitoring point S4, and it appears cracks.

Under $NPSHa=1.4$ m, the evolution process of cavitation patterns, as shown in figure 8(b). When $t=T/4$, a large scale continuous cavitation bubbles attached to the blade suction side of the No.1 flow passage, the trailing edge of cavitation bubbles extends to the monitoring point M5; When the No.1 flow passage rotates along the direction of impeller rotation to $t=T/2$, $3T/4$, the trailing edge of cavitation bubbles is retracted; When $t=T$, the trailing edge of cavitation bubbles growths towards the flow passage middle, and its volume expands to the monitoring point M4.

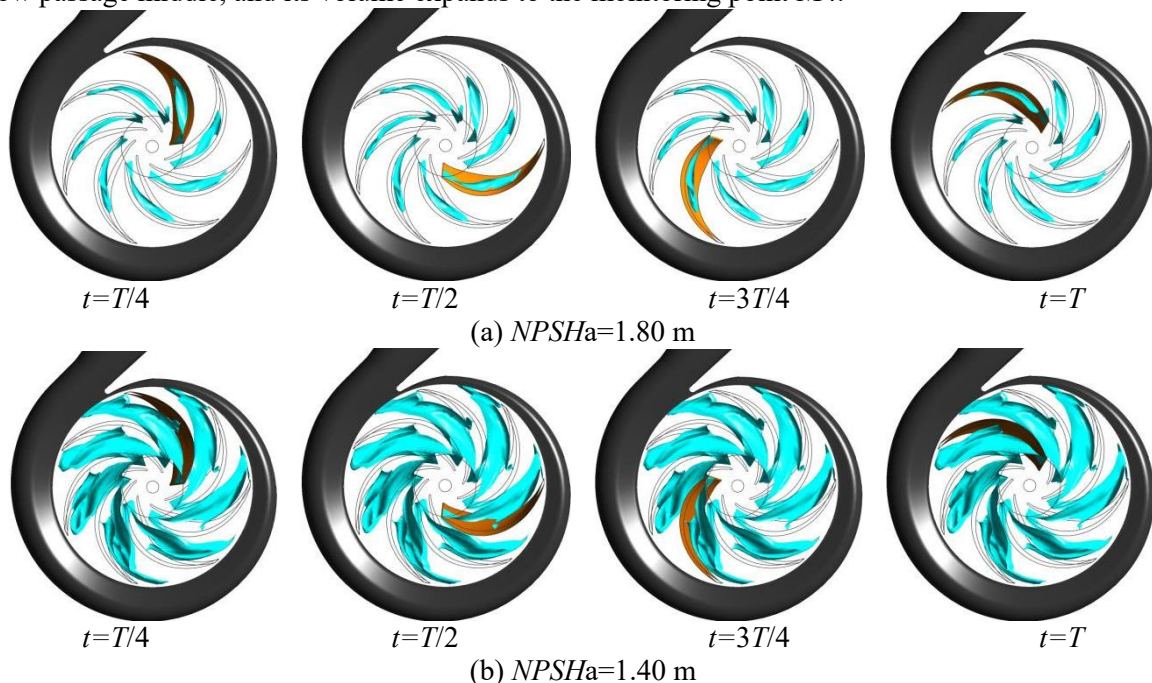


Figure 8. The evolution process of cavitation patterns in impeller.

4. Conclusion

In order to investigate the pressure fluctuation characteristics induced by cavitation in a centrifugal pump, the RNG $k-\varepsilon$ turbulence model and improved mass transport cavitation model has been

employed to calculate the unsteady cavitation flows. The numerical simulation is verified by comparing with experiment data.

Under non-cavitation, the dominant frequency of pressure fluctuation is impeller rotating frequency f_i ; Under $NPSHa=1.80$ m, the dominant frequency is f_i and $2f_i$; Under $NPSHa=1.40$ m, the dominant frequency is f_i .

Under non-cavitation, the maximum value of pressure fluctuation amplitudes in impeller appears at outlet; Under $NPSHa=1.80$ m, the maximum value in flow passage and blade pressure side appears at outlet, but in blade suction side it appears at 4/5 blade length from blade inlet, which about 1.6 times of non-cavitation; Under $NPSHa=1.40$ m, the maximum value in blade suction side and blade pressure side appears at outlet, but in flow passage it appears at 4/5 blade length from blade inlet, which about 2.2 times of non-cavitation.

The shedding and collapsing of cavitation bubbles in impeller flow passage, is resulting in the maximum value of pressure fluctuation amplitudes appears near 4/5 blade length from blade inlet.

Acknowledgments

This work was supported by the Inner Mongolia Higher School Scientific Research Program [Grant number NJZY17450].

References

- [1] Lin Z, Ruan X D, Zou J and Fu X 2014 *Chinese J Mech Eng* **27**(1) 165-170
- [2] Yao Z F, Wang F J, Xiao R F and He C L 2012 *Pro. of the 26th IAHR Sym. on Hydraulic Machinery and Systems*(Beijing, China, August19-23, 2012)
- [3] Chen T, Sun Y B, Liu Y L, Wu D Z and Wang L Q 2013 *Pro. of the 6th Int. Conf. on Pumps and Fans with Compressors and Wind Turbines*(Beijing, China, September19-22, 2013)
- [4] Tan L, Zhu B S, Wang Y C, Cao S L and Gui S B 2015 *Eng Computation* **32**(6) 1549-66
- [5] Xu Y, Tan L, Liu Y B and Cao S L 2017 *Adv Mech Eng* **9**(4) 1-12
- [6] Tan L, Zhu B S, Cao S L, Wang Y C and Wang B B 2014 *Energies* **7**(2) 1050-65
- [7] Cernetic J 2009 *P I Mech Eng C-J Mec* **223**(7) 1645-1655
- [8] Duplaa S, Coutier-Delgosha O, Dazin A, Roussette O, Bois G and Caignaert G 2010 *J. Fluids Eng.* **132** 021301-1
- [9] Ding H, Visser F C, Jiang Y and Furmanczyk M 2011 *J. Fluids Eng.* **133** 011101-1-14
- [10] Shi W D, Wang C, Wang W and Pei B 2014 *Adv Mech Eng* 367631
- [11] Liu H L, Liu D X, Wang Y, Wu X F, Wang J and Du H 2013 *J. of Hydrodynamics* **25**(3) 370-378
- [12] Zwart P J, Gerber A G and Belamri T 2004 *Pro. of the 5th ICMF Int. Conf. on Multiphase Flow*(Yokohama, Japan, May30-June3, 2004)
- [13] Meng G Q Q G, Tan L, Cao S L, Wang Y C, Xu Y and Qu W S 2014 *Pro. of the 6th Int. Sym. on Fluid Machinery and Fluids Engineering*(Wuhan, China, October22-25, 2014)