Numerical simulation and analysis of cavitation flows in a double suction centrifugal pump

G Meng1,2, L Tan1, S L Cao1,4, W Jian3, W W Liu3 and D J Jiang3
1 State Key Laboratory of Hydroscience and Engineering, Tsinghua University, Beijing 100084, China
2 Inner Mongolia Vocational College of Chemical Engineering, Huhehaote 010070, Inner Mongolia, China
3 Shanghai Eastpump (GROUP) CO., LTD, Shanghai 201906, China
E-mail: caoshl@mail.tsinghua.edu.cn

Abstract. Cavitation is an unsteady phenomenon, which is nearly inevitable in pumps. It would degrade the pump performance, generate vibrations and noises, and even erode pump flow passage components. The double suction centrifugal pump at design flow rate and large flow rate is numerically simulated using the \( k-\omega \) turbulence model and the mass transport cavitation model. As a result, the calculated variation of pump head with pump inlet pressure agreed well with the experimental data. The results demonstrate that the numerical model and method can accurately predict the cavitation flows in a double suction centrifugal pump. The cavitation characteristics are analysed in great details. In addition, based on the calculation results, the reason that the plunge of pump head curve is revealed. It is found that the steep fall of pump head happens when the cavity reaches the blade to blade throat and the micro-vortex group appears at the back of the blade suction side. At the same time, this practice can provide guidance for the optimal design of double suction pumps.

1. Introduction

For pumps, cavitation is a very important physical phenomenon. Cavitation bubbles emerge in pump operation, where the local pressure drops below the liquid vapor pressure [1]. Cavitation has great influence on the operation performance of pumping station unit, so that many problems have occurred, such as performance degradation, vibrations, noises, pump flow passage components erosion and so on. With the rapid development of the computer technology and computational fluid dynamics (CFD), numerical simulation has become one of the most important research methods for studying cavitation flow in pumps [2-5]. Tan et al. [6] simulated the cavitation flow of centrifugal pump by using the full cavitation model and modified \( RNG k-\varepsilon \) turbulence model, and analysed the cavitation characteristics inside the impeller and the pressure distribution at the impeller inlet. Pouffary et al. [7] accomplished cavitation simulation on a centrifugal pump based on the cavitation model of the barotropic state law and the standard \( k-\varepsilon \) turbulence model. When sheet cavitation passes through the narrowest section between two adjacent blades, the pump head drops obviously and the torque decreases accordingly. At the same time, the vacuoles block the passage, which results in an uneven load inside the channel of the impeller. Chang et al. [8] adopted the \( SST k-\varepsilon \) turbulence model and homogeneous mixture

4 Corresponding author.
cavitation model to simulate the cavitation flow in a mixed-flow pump. The results show that the cavitation firstly appears at suction side of blade near the leading edge and then extend to the outlet of impeller with the further reduction of net positive suction head. The serious blockage due to cavity in the flow passages results in a sharp reduction in pump head.

In the present research, the cavitation flows in a double suction centrifugal pump was predicted by using the $k-\omega$ turbulence model and the mass transport cavitation model. The calculated results are compared with the experimental data for different operating points. From these results, the cavitation characteristics and the origin of the cavitation head drop are discussed. Meanwhile, the analysis of their mechanism is proposed.

2. Computational domain and mesh

The research object is an ES8-300KPS double suction centrifugal pump as shown in Figure 1.

![Figure 1. Double suction centrifugal pump.](image)

The main parameters of the double suction centrifugal pump are given as follows: the volume flow rate $Q_d=820 \text{ m}^3/\text{h}$, the rotational speed $n=1480 \text{ r/min}$, the head $H=64 \text{ m}$, the number of blades $Z=6$. The computational domain consists of suction casing, impeller, volute, prolongations for suction casing inlet and volute outlet as shown in Figure 2. The suction casing, impeller and volute use the tetrahedral mesh, and the extension of inlet and outlet use the hexahedral mesh. The grid independence has been verified, the coarsest mesh with 4,198,349 elements is employed for the following calculations. The mesh elements for each part are shown in table 1.

![Figure 2. Computation domain of the double suction centrifugal pump.](image)

| Computation domain | Suction casing | Impeller | Volute |
|--------------------|----------------|----------|--------|
| Mesh               | 2 003 664      | 1 084 701| 1 109 984|
3. Mathematical model and numerical algorithm
The fluid in the cavitation flow field for the pumps is considered as a homogeneous and compressible mixed medium of vapor and liquid. The continuity and momentum equations in the Cartesian coordinates are as follows:

$$\frac{\partial (\rho_m u_i)}{\partial x_i} = 0$$  \hspace{1cm} (1)

$$\frac{\partial (\rho_m u_i u_j)}{\partial x_j} = - \frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j} \left[ (\mu_m + \mu_t) \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} - \frac{2}{3} \delta_{ij} \frac{\partial \delta}{\partial x_k} \right) \right]$$  \hspace{1cm} (2)

where \(\rho_m\) and \(\mu_m\) are the mixture of density and dynamic viscosity, calculated by weighted average of each phase volume fraction, \(u\) is the velocity, \(p\) is the pressure, and \(\mu_t\) is the turbulent viscosity, respectively. Subscripts \(i, j\) denote the axes directions. In the present study, the \(k-\omega\) turbulence model is selected.

The mass transport cavitation model was considered in this paper, which is deduced from the Rayleigh-Plesset equation:

$$\frac{\partial (\rho_v \alpha_v u_i)}{\partial x_i} = m^+ - m^-$$  \hspace{1cm} (3)

$$m^+ = C_{\text{vap}} \frac{3r_g (1-\alpha_v) \rho_v}{R_b} \sqrt{\frac{2 \max (p_v - p_c, 0)}{\rho_v}}$$  \hspace{1cm} (4)

$$m^- = C_{\text{cond}} \frac{3\alpha_v \rho_l}{R_b} \sqrt{\frac{2 \max (p - p_v, 0)}{\rho_l}}$$  \hspace{1cm} (5)

where \(\alpha_v\) is the vapor volume fraction, \(m^+\) and \(m^-\) represent the source terms for evaporation and condensation. \(C_{\text{vap}}\) and \(C_{\text{cond}}\) are the empirical calibration coefficients of evaporation and condensation, respectively. And \(r_g\) is the nucleation site volume fraction, \(R_b\) stands for the nucleation site radius. \(\rho_v\) is the vapor density, \(\rho_l\) is the liquid density, \(p_v\) is the water vaporization pressure. The recommended values of these coefficients are: \(C_{\text{vap}}=50\), \(C_{\text{cond}}=0.01\), \(r_g=5\times10^{-4}\), \(R_b=10^{-6}\) m, \(\rho_v=0.554\) kg/m\(^3\), \(\rho_l=997\) kg/m\(^3\) and \(p_v=3574\) Pa.

All the simulations were conducted by using the ANSYS-CFX commercial software. The boundary conditions are as follows: the total pressure at the inlet and the mass flow rate at the outlet based on the experimental measurements, the no-slip condition for the boundary layers imposed over the impeller blades and sidewalls, the volute casing and the inlet and outlet pipe walls. The liquid flow fields in the double suction centrifugal pump are solved first. Coupled with the cavitation model, the cavitation flow fields in the double suction centrifugal pump are solved by taking the liquid flow fields result as the initial condition. The pressure at the inlet is gradually reduced when the calculation is converged at a given pressure value.

4. Results and analysis

4.1. Pump performance curve and cavitation characteristics curve
Figure 3 shows the comparison of pump head and efficiency between the experimental and numerical results. There is a good quantitative agreement at the flow rate of 480–980 m\(^3\)/h, with the prediction errors smaller than 5.0%, which demonstrates that the numerical model and method can accurately predict the pump performance.
The net positive suction head available (NPSHa) is the difference between the fluid energy of unit weight in the entrance section of the computation domain and the energy of vaporization pressure. It is defined as

\[
NPSHa = \frac{p_{in}}{\rho g} + \frac{u_{in}^2}{2g} - \frac{p_{v}}{\rho g}
\]  

Figure 4(a) and 4(b) shows the experimental and numerical results of the head varying with NPSHa at 1.0Qd and 1.2Qd respectively. The pump head have large fluctuation at 1.2Qd condition, but the trend of numerical results still coincides with the experimental data.

4.2. The cavity development in the pump

Figure 5 and Figure 6 shows the cavity development in impeller with different NPSHa, corresponding to the letters A, B and C as shown in Fig.4(a) and 4(b), respectively. The cavity inception and development in the pump are clearly obtained. The cavity first appears on the suction side near the blade leading edge, as shown in Fig.5(a), at which condition there is no significant drop of the pump head. With the decrease of NPSHa, a large area of cavity emerges in the impeller channels and expands to the pressure side. Hence, the pump normal flow will be affected, and the pump head begins to drop, as shown in Fig.5(b). In Figure 5(c), when NPSHa=3.17 m, the impeller channels are seriously blocked by the cavities, which results in the pump head breaking down. On the other hand, in terms of the cavity inception and development, same tendency can be seen in large flow rate, as shown in Fig.6.
A section located at $z=0.08$ m is selected to analyse the pressure distribution of the impeller inlet. Figure 7 and Figure 8 present the distributions of pressure at the impeller inlet. With the reduction of $NPSH_a$, the pressure at the impeller inlet decreases. As the pressure gradient becomes smaller, the pressure distribution becomes more even.

4.3. Internal flow characteristics analysis
Figure 9 shows the streamlines on middle span section of impeller with different NPSHa at 1.0Qd. The streamline distribution is relatively uniform, as shown in Figure 9(a). With the decrease in NPSHa, the streamlines disorder at the blade suction side near the trailing edge, as shown in Figure 9(b). In Figure 9(c), the streamlines at the back of the blade suction side is disorder and generate the vortex micro group. At the same time, the velocity is uneven at impeller outlet, with the decrease in NPSHa, the low velocity area expands gradually, as shown in Figure 10, which induces the instability of pump operation.

![Streamlines](image1)

(a) NPSHa=6.00 m  (b) NPSHa=3.57 m  (c) NPSHa=3.17 m  

**Figure 9.** Streamline on middle span section of impeller at 1.0Qd.

![Velocity Distribution](image2)

(a) NPSHa=6.00 m  (b) NPSHa=3.57 m  (c) NPSHa=3.17 m  

**Figure 10.** Velocity distribution at outlet of impeller for 1.0Qd.

5. Conclusions
The cavitation flow in a double suction centrifugal pump is calculated by using the $k-\omega$ turbulence model and the mass transport cavitation model. Based on the results, concluding remarks as follows were obtained.

1) The calculated results of pump head varying with net positive suction head available are consistent with the measurements, which demonstrates that the numerical model and method can accurately predict the cavitation flows in a double suction centrifugal pump.

2) For the design flow rate and large flow rate, the cavity inception and development in the pump is similar.

3) The steep fall of pump head is caused when the cavity reaches the blade to blade throat and the vortex micro group appears at the back of the blade suction side.

References
[1] Ding H, Visser F C, Jiang Y and Furmanczyk M 2011 *J. Fluid Eng.* **133**(1) 011101
[2] Tan L, Cao S L, Wang Y M and Zhu B S *Transactions of the Chinese Society of Agricultural Machinery* 2012 **43**(9) 49-52 (in Chinese)
[3] Ding H, Visser F C and Jiang Y 2012 A practical approach to speed up NPSHr prediction of centrifugal pumps using CFD cavitation model *ASME 2012 Fluids Engineering Division Summer Meeting collocated with the ASME 2012 Heat Transfer Summer Conference and the ASME 2012 10th International Conference on Nanochannels, Microchannels, and
Minichannels (Rio Grande, Puerto Rico, 8-12 July 2012)

[4] Chen T, Li S R, Li W Z, Liu Y L, Wu D Z and Wang L Q 2013 IOP Conference Series: Materials Science and Engineering 52(6): 062020

[5] Tan L, Zhu B S, Cao S L and Wang B 2014 Energies 7(2): 1050-65

[6] Tan L, Cao S L, Wang Y M and Zhu B S 2012 Chinese Phys. Lett. 29(1) 14702-4

[7] Pouffary B, Patella R F, Reboud J L and Lambert P A 2008 J. Fluid Eng. 130(6) 061301

[8] Chang S P and Wang Y S 2012 J. of Drainage and Irrigation Machinery Eng. 30(2): 171-5 (in Chinese)