Numerical investigation of unsteady turbulent flow in a centrifugal pump at partial load

T Lei, Z Baoshan, C ShuLiang, W Yuchuan, W Xuhe
State Key Laboratory of Hydrosience and Engineering, Tsinghua University, Beijing, 100084, China
E-mail: bszhu@mail.tsinghua.edu.cn

Abstract. The unsteady non-cavitation and cavitation turbulent flows in a centrifugal pump at partial load condition are numerically investigated by CFX 13.0. The numerical framework employs the combination of RNG k-ε turbulence model and transport equation cavitation model, in which the effects of compressibility of fluid on cavitation region and pressure fluctuation on saturation pressure are both taken into consideration. The good agreement between the numerical and experimental values validates that the numerical framework can accurately predict the turbulent flows in the centrifugal pump. The complex flow characteristics in impeller at non-cavitation and cavitation conditions are revealed. For the non-cavitation flow, the dominant frequencies of pressure fluctuation of monitoring points in impeller are all the Impeller Rotation Frequency 24.17Hz. The maximum value of pressure fluctuation on the blade pressure side appears at the 0.8 chord length from the blade leading edge due to a clockwise rotating vortex, which incepts, develops and disappears when the corresponding blade passes through the volute tongue. The dominant frequencies of pressure fluctuation of monitoring points in volute are the Blade Pass Frequency 145 Hz or twice of it. The maximum value of pressure fluctuation in the volute appears near the tongue region, where the flow fields are uneven with strong second flow in the cross section. For the cavitation flow, as the cavitation develops at the blade leading edge, the turbulent flows in the impeller are greatly influenced by the bubble shedding and collapse. The maximum values of pressure fluctuation in impeller increase with the development of cavitation, and reach the largest magnification of about 2.0 in comparison to the non-cavitation flow when the pressure at the pump inlet is very low. The complicated phenomenon of unsteady turbulent flow in a centrifugal pump indicates that the vortex has great influence on the flow pattern.

1. Introduction
Due to the asymmetric shape of the spiral volute and tongue, the flow in the impeller is strongly interacting with the volute flow, and the flow pattern is extremely complicated when the centrifugal pump works at partial load condition. Cavitation is a common physical phenomenon in centrifugal pump, which induces pressure fluctuation and uneven load distribution, and then significantly reduces the service life of centrifugal pump [1, 2].

Many studies have investigated the unsteady turbulent flow field in centrifugal pumps. Arndt et al. [3] firstly reported the unsteady pressure measurement of both impeller blade and diffuser vane in a diffuser pump by experiments. Parrondo et al. [4] conducted a systematic series of experiments to
measure the pressure fluctuations on 36 points around the volute of a centrifugal pump for flow rates of 0% to 160% of the best efficiency operation. The experimental data indicate that the pressure fluctuations result from the superposition of the hydraulic disturbances and the interaction of blade and volute tongue. Majidi [5] solved the unsteady three dimensional viscous flows in a centrifugal pump to obtain the unsteady pressure distribution in the impeller and volute. Due to the interaction between impeller and volute, the pressure fluctuations are strong at impeller outlet and at the vicinity of the tongue. Pavesi et al. [6] carried an experimental investigation of the flow field instability in a centrifugal pump. The results highlighted the existence of an asymmetrical rotating pressure structure at the impeller outlet, and a fluid-dynamical origin and propagating both in the radial and circumferential direction appears to be connected with the phenomenon of jet-wake. Barrio et al. [7] completed numerical analysis of the unsteady flow in the near-tongue region in a volute-type centrifugal pump for several operating conditions ranging from 20% to 160% of the nominal flow rate. LI [8] investigated the pressure and velocity pulsations in two centrifugal pumps with different blade exit angles and liquids by means of experiment and numerical simulation. The results show that a larger fluid viscosity and a smaller blade exit angle can reduce the fluctuation of flow variables.

The cavitation phenomenon and mechanism in a pump has been studied in recent years. Hosangadi et al. [9] used an acoustically accurate, compressible multiphase model to simulate a cavitation inducer at design flow conditions with different inflow pressures. The computational results indicated that the loss in performance directly correlates with the amount of blockage in the blade flow passages caused by cavitation. Cervone et al. [10] set up an optical system in order to investigate the cavitation flow instabilities on a three-bladed inducer in the cavitating pump. The tip cavity length and the blade cavitating areas are calculated and analysed. Kimura et al. [11] conducted 3D unsteady computational fluid dynamics simulations of cavitating flow in a turbopump inducer for three types of inlet casing geometry with various flow. It was found that the tip leakage vortex was dependent on the inlet casing geometry and the flow rate, and was responsible for the appearance of rotating cavitation. Bachert et al. [12] took PIV measurement to investigate the cavitation flow for the case of 3% head drop in a centrifugal pump operating at overload conditions of 1.17 \( Q_d \). The study revealed that the 3% head drop was caused by the cavitation appearance on the volute tongue while almost no cavitation was present on the impeller blades.

Numerous research on the unsteady turbulent flow in a centrifugal pump has been conducted, but the flow characteristics at partial load condition is still difficult to capture accurately, especially when the cavitation occurs. In this paper, the unsteady turbulent flow without and with cavitation in a centrifugal pump at partial load condition is numerical simulated to reveal the flow field in impeller and volute.

2. Mathematical Model and Numerical Algorithm

The fluid in the cavitation flow field is considered a homogeneous and compressible mixed medium of liquid and vapour. The continuity and momentum equations in the Cartesian coordinates are as follows:

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

\[
\frac{\partial (\rho_m u_i)}{\partial t} + \frac{\partial (\rho_m u_i u_j)}{\partial x_j} = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j} \left[ (\mu_m + \mu_t) \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} - \frac{2}{3} \frac{\partial u_k}{\partial x_k} \delta_{ij} \right) \right]
\]  

where \( \rho \) and \( \mu \) are respectively the density and dynamic viscosity, calculated by the weighted average of each phase volume fraction \( \alpha \); \( \rho_m = \rho_l \alpha_l + \rho_v \alpha_v \), \( \mu_m = \mu_l \alpha_l + \mu_v \alpha_v \); subscripts \( l \) and \( v \) denote the liquid phase, the vapour phase and the mixture, respectively; \( u \) is the velocity; \( p \) is the pressure; \( \mu_t \) is the turbulent viscosity. Subscripts \( i, j \) and \( k \) denote the axis directions.

The RNG \( k-\varepsilon \) turbulence model is used by modifying the turbulent viscosity with a density function [13]. The liquid-vapour mass transfers due to cavitation are solved by the transport equation-based cavitation model.

.. figure:: 2.png

   | Caption | Description |
   |--------|-------------|
   | Figure 2 | Integration of the RNG \( k-\varepsilon \) turbulence model |

2
\[
\frac{\partial \left( \rho \alpha_e \right)}{\partial t} + \nabla \cdot \left( \rho \alpha_e u \right) = \dot{m}_{vap} - \dot{m}_{con}
\]  
(3)

in which the mass transfers for vapourization and condensation rates proposed by Zwart et al. [14] are modelled as

\[
\dot{m}_{vap} = C_{vap} \frac{3 \alpha_{nuc} (1 - \alpha_e) \rho_v}{R_b} \sqrt{\frac{2 \max \left( p_v - p, 0 \right)}{\rho_l}}
\]  
(4)

\[
\dot{m}_{con} = C_{con} \frac{3 \alpha_e \rho_v}{R_b} \sqrt{\frac{2 \max \left( p - p_v, 0 \right)}{\rho_l}}
\]  
(5)

where \(C_{vap}\) and \(C_{con}\) are the empirical calibration coefficients for vapourization and condensation rates, respectively; \(R_b\) is the bubble radius; \(p_v\) is the vapour pressure; \(\alpha_{nuc}\) is the nucleation site volume fraction. The vapour pressure \(p_v\) in this cavitation model is modified by taking consideration of the influence of local turbulent pressure fluctuation:

\[
p_v = p_{sat} + 0.195 \rho_m k
\]  
(6)

where \(p_{sat}\) is the saturation pressure at the test temperature.

The computation fluid dynamics code CFX 13.0 is employed to the combination of RNG \(k-\varepsilon\) turbulence and transport equation-based cavitation model. The boundary conditions are defined as: the total pressure at pump inlet, and the mass flow at the pump outlet, no slip wall at walls. The results of the steady calculation are taken as the initial flow field in unsteady flow calculation.

3. Problem Statement and Numerical Parameters

3.1. Geometry and mesh of centrifugal pump

The centrifugal pump used in the present investigation is a conventional single suction centrifugal pump with a shrouded impeller and six backswept blades. The main parameters of the pump are listed in Table 1. The partial load condition at discharge \(Q = 0.76Q_d = 260 \text{ m}^3/\text{h}\) is chosen for the numerical investigation.

| Parameter                  | Value  |
|---------------------------|--------|
| Rated discharge \(Q_d\) (m$^3$/h) | 340    |
| Rated head \(H\) (m)       | 30     |
| Rotational speed \(n\) (r/min) | 1450   |
| Number of blade \(Z_i\)    | 6      |
| Impeller diameter \(D_2\) (mm) | 329    |
| Blade width at exit \(b_2\) (mm) | 34.6   |

Figure 1(a) shows the whole passage of the computation domain including the suction pipe, impeller and volute. Structural hexahedral meshes are used in the whole passage of centrifugal pump, and the meshes near the blade surface and volute tongue are locally refined to get better calculation accuracy. As shown in Fig. 2(b) and (d), 15 monitoring points in impeller and 14 monitoring points in volute are set at the middle span of the centrifugal pump.
3.2. Mesh density and time step independences

Five sets of meshes with elements from 1,804,742 to 5,083,916 are chosen to test the mesh independence as shown in Fig. 2. The result shows that the pump head and efficiency vary tiny for the five mesh densities, therefore the coarsest mesh with 1,804,742 elements is employed in the present calculation.

Three sets of time steps $\Delta t=1.08 \times 10^{-4}$, $2.15 \times 10^{-4}$ and $4.31 \times 10^{-4}$ s, are chosen to test the time step independence. The monitored pressures on monitoring points PM2, PS2 and SS2 in Fig. 3 show little differences for different time steps, therefore $\Delta t=2.15 \times 10^{-4}$ s is employed in the present calculation.
3.3. Validation of calculation results

To evaluate the cavitation performance of pumps, the net positive suction head available (NPSHA) is defined in Eq. (7), which is the difference between the total energy and vaporization energy for the unit weight of the fluid at pump inlet.

$$NPSHA = \frac{p}{\rho g} + \frac{u^2}{2g} - \frac{p_v}{\rho g}$$  \hspace{1cm} (7)

The cavitation performance of the calculated pump is tested by using laboratory measurements in previous studies [1, 12, 13]. As shown in Fig.4, the numerical simulated head-NPSHA curve of the centrifugal pump is in good agreement with the experiment data, especially the sudden decreasing tendency around the critical value.
4. Results and Discussion

4.1. Development of cavitation in impeller
Figure 5 shows the development of cavitation in impeller as the pressures gradually drop at pump inlet according to the measurement data. First, there is no cavitation phenomenon in the impeller when the NPSHA is large enough. At the NPSHA = 2.2 m, the cavitation develops to occupy the blade leading edge and make the pump head begin to decrease as shown in Fig.4. As the NPSHA decreases further to 1.5 m, the cavitation develops fully in the impeller.

4.2. Time evolution of pressure in impeller and volute
Figure 6 and 7 show the time evolution of pressure in pump impeller and volute at non-cavitation condition. The pressures in middle passage of impeller gradually increase from impeller inlet PM1 to outlet PM5 due to the input shaft power. Meanwhile, the amplitudes of pressure fluctuation also magnify from PM1 to PM5. In the centrifugal pump volute, the pressures increase along the movement direction of fluid from V2 to V11, because the function of volute is to transform the fluid kinetic energy to pressure energy.
4.3 Frequency characteristics of pressure fluctuations in impeller and volute

Figure 8 shows the frequency domain of pressure in impeller at non-cavitation condition. For the non-cavitation condition, the dominant frequencies of pressure fluctuation of monitoring points in impeller are all the Impeller Rotation Frequency $f_i = n/60 = 24.17$ Hz.

**Figure 6.** Time evolution of pressure on PM1-PM5 in impeller.

**Figure 7.** Time evolution of pressure on V2, V5, V8 and V11 in volute.
Figure 8. Frequency domain of pressure in impeller.

Table 2 lists the maximum amplitudes of pressure fluctuation in impeller at non-cavitation and cavitation conditions. The maximum amplitudes of pressure fluctuation increase in the impeller from inlet to outlet. However, for the non-cavitation condition, the maximum amplitude of pressure fluctuation on monitoring point PS4 is larger than that of other points. For the cavitation flow, as the cavitation develops at the blade leading edge, the turbulent flows in the impeller are greatly influenced by the bubble shedding and collapse. The maximum values of pressure fluctuation in impeller increase with the development of cavitation, and reach the largest magnification of about 2.0 in comparison to the non-cavitation flow when the pressure at the pump inlet is very low with NPSHA=1.5 m.

### Table 2. Pressure fluctuations amplitude in the impeller.

| Monitoring point | Maximum amplitudes of pressure fluctuation (pa) | Non-cavitation | NPSHA=2.2 m | NPSHA=1.5 m |
|------------------|-----------------------------------------------|----------------|-------------|-------------|
| PM1              | 3247                                          | 5154           | 7275        |
| PM2              | 8277                                          | 9966           | 10664       |
| PM3              | 14969                                         | 16225          | 17883       |
| PM4              | 20021                                         | 21614          | 21802       |
| PM5              | 23265                                         | 25479          | 26914       |
| PS1              | 7280                                          | 8722           | 13139       |
| PS2              | 12096                                         | 14373          | 15771       |
| PS3              | 19401                                         | 21569          | 21809       |
| PS4              | 27843                                         | 29158          | 29825       |
| PS5              | 26584                                         | 29430          | 31333       |
| SS1              | 3080                                          | 0              | 0           |
| SS2              | 4087                                          | 4643           | 10373       |
| SS3              | 11762                                         | 15207          | 16182       |
| SS4              | 17868                                         | 21807          | 21907       |
| SS5              | 3247                                          | 5154           | 7275        |

The dominant frequencies of pressure fluctuation of monitoring points in volute are the Blade Pass Frequency $f_{BPF}=n^*z_i/60=1450^*6/60=145$ Hz or twice of $f_{BPF}$. Table 3 lists the maximum amplitudes of...
pressure fluctuation in volute at non-cavitation condition. The maximum value of pressure fluctuation in the volute appears at the monitoring point V2.

### Table 3. Frequency characteristics in volute.

| Monitoring point | Frequency (Hz) | Maximum amplitudes of pressure fluctuation (pa) |
|------------------|----------------|-----------------------------------------------|
| V2               | 145            | 10372                                         |
| V5               | 145            | 2484                                          |
| V8               | 145            | 3472                                          |
| V11              | 290            | 3408                                          |

### 4.4. Flow field in impeller and volute

Figure 9 shows the pressure distribution and vector map in impeller and volute at the instantaneous time of $t=0.5086 \text{ s}$ in the transient calculation. The pressure distribution in impeller is asymmetrical due to the circumferential distortion of volute geometry and the interaction between the rotational impeller and stable volute. The pressure gradient in the volute tongue region is relatively greater than that of other region in the volute. Therefore, the maximum amplitude of pressure fluctuation on monitoring point V2 is the largest of all monitoring points in the volute. There is a clockwise vortex located close to the monitoring point PS4, about 0.8 chord length from the blade leading edge, as shown in Figs. 9(b). With the corresponding blade passing through the volute tongue, the clockwise vortex incepts, develops and disappears at this local region, which induces a strong pressure pulsation at monitoring point PS4.

![Figure 9. Instantaneous flow field at $t=0.5086 \text{ s}$: (a) pressure distribution; (b) vector map.](image)

Figure 10 shows the streamlines at cross sections V2, V5, V8 and V11 in the volute. The flow fields on all the four cross sections are uneven with strong second flow. The intensity of second flow along flow direction from V2 section to V11 section gradually strengthens, because the radial pressure gradient gradually increases. On the V8 section, the single vortex evolves as a counter-rotating vortex pair. This counter-rotating vortex pair is asymmetric in vortex structure and strength.
5. Conclusions
A computation model combining the modified RNG $k$-$\epsilon$ turbulence model and transport equation cavitation model is employed in commercial code CFX 13.0 to numerically simulate the unsteady turbulent flow in the centrifugal pump. The satisfactory agreement between the numerical and experimental results of the pump head varying with the NPSHA shows that the numerical simulation has the capability to calculate the flow field in the centrifugal pump.
In the impeller, the dominant frequencies of pressure fluctuation are all the Impeller Rotation Frequency 24.17Hz. While in the volute, the dominant frequencies of pressure fluctuation are the Blade Pass Frequency 145 Hz or 290 Hz depending on the location in the volute. The maximum value of pressure fluctuation in the impeller appears on the monitoring point PS4 due to a time-varying vortex located in this region. The flow fields in the volute are uneven with strong second flow in the cross section. The pressure fluctuation research maximum at the V2 section, because the area of this section is narrow and the velocity is relatively high. The cavitation has great influence on the flow characteristic in centrifugal pump.

6. References
[1] Tan L, Zhu B S, Cao S L, et al. Influence of Prewhirl Regulation by Inlet Guide Vanes on Cavitation Performance of a Centrifugal Pump. Energies. 2014, 7(2): 1050-1065.
[2] Tan L, Zhu B S, Cao S L, et al. Cavitation flow simulation for a centrifugal pump at a low flow rate. Chinese. Sci. Bull. 2013, 58(8): 949-952.
[3] Arndt N, Acosta A, Brennen C, et al. Experimental investigation of rotor-stator interaction in a centrifugal pump with several vaned diffusers. *J. Turbomach.* 1990, 112(1): 98-108.

[4] Parrondo-Gayo J L, Gonzalez-Perez J, Fernandez-Francos J. The effect of the operating point on the pressure fluctuations at the blade passage frequency in the volute of a centrifugal pump. *J. Fluids Eng.* 2002, 124(3): 784-790.

[5] Majidi K. Numerical study of unsteady flow in a centrifugal pump. *J Turbomach,* 2005, 127(4): 363-371.

[6] Pavesi G, Cavazzini G, Ardizzo G. Time-frequency characterization of the unsteady phenomena in a centrifugal pump. *Int. J. Heat. Fluid. Fl.* 2008, 29(5): 1527-1540.

[7] Barrio R, Parrondo J, Blanco E. Numerical analysis of the unsteady flow in the near-tongue region in a volute-type centrifugal pump for different operating points. *Comput. Fluids.* 2010, 39(5): 859-870.

[8] Li W G. Effects of blade exit angle and liquid viscosity on unsteady flow in centrifugal pumps. *Proc. Inst. Mech. Eng. Part A J. Power Energy.* 2012, 226(4): 580-599.

[9] Hosangadi A, Ahuja V, Ungewriter R. Simulations of cavitating flows in turbopumps. *J. Propul. Power.* 2004, 20(4): 604-611.

[10] Cervone A, Bramanti C, Torre L, et al. Setup of a high-speed optical system for the characterization of flow instabilities generated by cavitation. *J. Fluids Eng.* 2007, 129(7): 877-885.

[11] Kimura T, Yoshida Y, Hashimoto T, et al. Numerical Simulation for Vortex Structure in a Turbopump Inducer: Close Relationship With Appearance of Cavitation Instabilities. *J. Fluids Eng.* 2008, 130(5).

[12] Bachert R, Stoffel B, Dular M. Unsteady cavitation at the tongue of the volute of a centrifugal pump. *J. Fluids Eng.* 2010, 132: 061301.

[13] Coutier-Delgosha O, Fortes-Patella R, Reboud J. Evaluation of the turbulence model influence on the numerical simulations of unsteady cavitation. *J. Fluids Eng.* 2003, 125: 38-45.

[14] Zwart P J, Gerber A G, Belamri T. A two-phase flow model for predicting cavitation dynamics. Proc of Fifth International Conference on Multiphase Flow, Yokohama, Japan. 2004.

[15] Tan L, Cao S L, Gui S B. Hydraulic design and pre-whirl regulation law of inlet guide vane for centrifugal pump. *Sci. China. Tech. Sci.* 2010, 53(8): 2142-2151.

[16] Tan L, Cao S L, Wang Y M, et al. Influence of axial distance on pre-whirl regulation by the inlet guide vanes for a centrifugal pump. *Sci. China. Tech. Sci.* 2012, 55(4): 1037-1043.

**Acknowledgments**

This work was supported by the National Natural Science Foundation of China [Grant number 51176088 and 51179090].