Research on the performance of low-lift diving tubular pumping system by CFD and Test

Chenzhi Xia, Li Cheng*, Chao Liu, Jiren Zhou, Fangping Tang and Yan Jin

School of Hydraulic, Energy and power Engineering
Yangzhou University, Yangzhou, Jiangsu, P.R.China, 225009

E-mail: chengli@yzu.edu.cn

Abstract. Post-diving tubular pump is always used in large-discharge & low-head irrigation or storm drainage pumping station, its impeller and motor share the same shaft. Considering diving tubular pump system’s excellent hydraulic performance, compact structure, good noise resistance and low operating cost, it is used in Chinese pump stations. To study the hydraulic performance and pressure fluctuation of inlet and outlet passage in diving tubular pump system, both of steady and unsteady full flow fields are numerically simulated at three flow rate conditions by using CFD commercial software. The asymmetry of the longitudinal structure of inlet passage affects the flow pattern on outlet. Especially at small flow rate condition, structural asymmetry will result in the uneven velocity distribution on the outlet of passage inlet. The axial velocity distribution uniformity increases as the flow rate increases on the inlet of passage inlet, and there is a positive correlation between hydraulic loss in the passage inlet and flow rate’s quadratic. The axial velocity distribution uniformity on the outlet of passage inlet is 90% at design flow rate condition. The predicted result shows the same trend with test result, and the range of high efficiency area between predicted result and test result is almost identical. The dominant frequency of pressure pulsation is low frequency in inlet passage at design condition. The dominant frequency is high frequency in inlet passage at small and large flow rate condition. At large flow rate condition, the flow pattern is significantly affected by the rotation of impeller in inlet passage. At off-design condition, the pressure pulsation is strong at outlet passage. At design condition, the dominant frequency is 35.57Hz, which is double rotation frequency.

1. Introduction
The low head axial flow pump is widely applied in large hydraulic engineering, i.e., the South-to-North Water Diversion Projects. Many researchers have focused on tip clearance, hub leakage, rotor-stator interaction and cavitation for axial-flow pump improvement. Zierke [1] proposed an experiment technology and discussed the flow characteristics of high Reynolds number pump and tip clearance flow. Zhang De-sheng et al. [2] used experiment and numerical method to analysis unsteady flow of axial-flow pump.
Post-diving tubular pump is application of axial-flow pump. Its pump is combined with motor so that the structure can be compact. The inlet and outlet passage are on the straight line, so the shape of tubular pump system looks like a cylinder. The head loss of tubular pump system is less than others. The efficiency of that is high too. Besides, the pump system also has other advantages such as the convenience of installation and overhaul. It is widely used in low-lift pump station, for example, there is seven new tubular pump station that have been built in the first phase of the east route of South-to-
North Water Diversion Project [3]. Compared with traditional bulb tubular pump, the diving pump use submarine motor instead of ordinary motors. The body of submarine motor is smaller than ordinary motor for that submarine motor reduces the number of unnecessary structures [4-7]. The number of medium and small pump stations that use the diving tubular pump system is gradually increased. Besides, the large pump station begin to use the diving tubular pump system. The production and application of large-diameter diving pump rapidly develop. Huangtang pump station installed eight 1800-calibre diving pumps in Meizhou Guangdong in 2007. Its design head is 3.6 m and design flow rate is 14 m$^3$/s. Chuzhou pump station uses six 2100-calibre diving pumps in Huai'an Jiangsu in 2013. Its design head is 1.3 m and the design flow rate of signal pump is 12.5 m$^3$/s. The maximum efficiency is 81% [8].

Many authors contributed to the research of pump performance both steady and unsteady by numerical method. Li Yaojun [9] carried out the numerical investigation on the interaction of flow through the inducer and impeller of an axial-flow pump equipped with an inducer. Zhou Ling [10] investigated a multistage deep-well centrifugal pump with different impeller rear shroud radius both numerically and experimentally under multiconditions. Zhang Ning [11] carried out an experimental investigation on unsteady pressure pulsation in a centrifugal pump with special slope solute.

2. Calculate model and numerical method

2.1. Model of pump system

GL-2008-03 model is used in this diving tubular pump system, which is tested in South-to-North Water Diversion Project [12]. The diameter of impeller is 300mm. The number of blades is three. The rotating speed is 1067 RPM. The angle of blade is 0°. The blade tip clearance is 0.15mm. Number of guide vanes is five. Figure 1 shows sketch of the diving tubular pump system. P1, P2 and P3 are the monitor points for pressure fluctuation in inlet passage. P4, P5 and P6 are the monitor points for pressure fluctuation in outlet passage.

![Fig.1 The sketch of 3-D model of diving tubular pump system](image)

2.2. Mesh

The mesh topology of impeller is H/J/L-Grid and the guide vanes’ mesh topology is H-Grid. Hexahedral mesh is applied on bulb, inlet and outlet passage in ICEM CFD. The table 1 shows the mesh number and quality of different parts of pump system. The quality of mesh has high influence on the simulating results about pump system. The calculation domain should not have one negative volume mesh elementary. The orthogonality of mesh is decided by the angle of any two surfaces of mesh elementary. The range of orthometric angle is between 10° and 165°. The picture of impellor’s and guide vanes’ mesh is showed in Fig.2.

| Parts    | Nodes | Orthogonality |
|----------|-------|---------------|
|          |       | Minimum angle | Maximum angle |
| Impeller | 558112 | 17            | 162           |
| Guide vane | 610140 | 19            | 160           |
2.3. **Calculate method**

The low-lift diving tubular pumping system was simulated based on Navier-Stoke solver embedded RNG k-ε turbulence model by commercial CFD software [13]. Both steady simulation and unsteady simulation were conducted in this paper. The interface between rotor and stator is set to frozen rotor in steady simulation and transient frozen rotor in unsteady simulation. The inlet boundary is set to massflow. The outlet boundary is set to outflow.

3. **Hydraulic performance**

The paper picks three special condition of pump system to analyse, which are low flow rate condition, design condition and large flow rate condition. The low flow rate condition is \( Q/Q_d = 0.667 \). The design condition is \( Q_d \), and large flow rate condition is \( Q/Q_d = 1.111 \). 

3.1. **Performance of inlet passage and outlet passage**

The inlet passage is designed to provide good and uniform flow pattern for axial flow pump. But the sections of inlet passage always shrink too fast to get good flow pattern in the outlet for the limit of space in actual engineering. Consequently, the performance of inlet passage has obvious influence on efficiency of pump system.

Note: Red plane named as 1-1 is cross section; grey plane named as 2-2 is vertical section; green plane named as 3-3 is outlet of inlet passage.

Fig. 4 Vertical section in inlet passage

Fig. 5 compares the distributions of axial velocity in inlet passage in different flow rate condition. Comparing with low flow rate condition, the axial velocity gradually increases along flow direction at cross section in design condition and large flow rate condition. Besides, the distribution of axial velocity
is almost symmetrical and the uniformity of axial velocity distribution is better. But the distribution of axial velocity at vertical section is different from that at cross section. The vertical section is divided to two area which are upper area and lower area by the centre line of impeller, which is showed in Fig. 4. The distribution is asymmetrical at vertical section for the structural asymmetry of inlet passage. The structural asymmetry leads to that the head loss in upper area of inlet passage is more than that in lower area, so that the axial velocity is lesser in upper area of outlet than that in lower area. Meanwhile, the negative axial velocity areas are found near outlet of inlet passage at vertical and cross section in low flow rate condition. That means that recirculation and vortex exist in low flow rate condition, so that flow pattern is bad for impeller.

Fig. 6 shows axial velocity distribution at inlet passage outlet in three special condition. There are three recirculation zones at inlet passage outlet in low flow rate condition, which means that the flow pattern is bad for pump operation. Besides, flow pattern is better and more uniform at the outlet in design condition and large flow rate condition, compared to low flow rate condition.

\begin{align*}
\text{Note: a, b and c are cross section; d, e and f are vertical section.}
\text{Fig. 5 Axial velocity contours}
\end{align*}

\begin{align*}
\text{Fig. 6 Axial velocity distribution at inlet passage outlet}
\end{align*}

This paper researches on performance of inlet passage with axial velocity distribution uniformity. The definition of axial velocity distribution uniformity is shown in Equation (1).

\begin{equation}
V_{u+} = \left[ 1 - \frac{1}{\bar{u}_n} \sqrt{\frac{\sum(u_{ai} - \bar{u}_a)^2}{n}} \right] \times 100\%
\end{equation}
Note: \( \bar{u}_a \) is mean axial velocity at outlet section 3-3, m/s; \( u_{ai} \) is axial velocity of each element at outlet section 3-3, m/s; \( n \) is the number of element divided for simulation at outlet section 3-3.

The result shows in Fig. 7. The uniformity is 65% at the inlet passage outlet in low flow rate condition. In design condition, the uniformity is 90%. In large flow rate condition, the uniformity is 91%. Axial velocity distribution uniformity becomes high and rate of the uniformity increasing becomes down with the increasing of flow rate.

![Fig. 7 Axial velocity distribution uniformity](image)

Fig. 7 Axial velocity distribution uniformity

Fig. 8 gives streamlines in outlet passage in three different conditions. Flow velocity decreases along the flow direction in outlet passage, while the cross section area increases. Recirculation and vortex are found in the inlet of outlet passage in low flow rate condition. Besides, the flow pattern become better in the outlet of outlet passage for the sufficient length to rectify.

![Fig. 8 Streamlines in outlet passage in different condition](image)

(a) 0.667 \( Q_D \)  (b) \( Q_D \)  (c) 1.111 \( Q_D \)

Fig. 8 Streamlines in outlet passage in different condition

This paper use head loss to evaluate the performance of outlet passage. Fig. 9 is the relation between head loss (\( \Delta h \)) and flow rate coefficient (\( Q/Q_D \)). Relation between head loss and flow rate is negative correlation at outlet passage, while \( Q/Q_D \) is between 0.556 and 1.037. That relation is positive correlation at outlet passage, while \( Q/Q_D \) is between 1.037 and 1.148. That means the performance of outlet passage is related to operation condition of pump system. The performance of outlet passage is better in design condition than others.

![Fig. 9 Head loss of outlet passage](image)

![Fig. 9 Head loss of outlet passage](image)

3.2. Pressure Fluctuations

Spectrum analysis can overcome the shortage of randomness of mathematical statics, which is widely used to analyse pressure pulsation by many scholars. Fast Fourier Transform (FFT) is applied to analyse
the unsteady pressure features based on data of unsteady simulation. Pressure coefficient $C_p$ is defined in equation (2).

$$C_p = \frac{p - p_0}{0.5 \rho v_d^2}$$

The rotation speed of impeller is 1067 r/min, which is transformed to 17.78 Hz presented by $f_r$. So $f_r$ is defined as the rotation frequency of impeller.

Table 2 presents frequency spectrum parameters of P1, P2 and P3. The dominant frequency at the three points is 355.7Hz in low flow rate condition, whose value is equal to $20f_r$. The dominant frequency is 35.57Hz in design condition, whose value is equal to $2f_r$. The dominant frequency is 551.3Hz, whose value is equal to $31f_r$. The main frequencies and the value of $C_p$ are same at three points in the same condition.

Comparing to off design condition, the dominant frequency of pressure fluctuation is low frequency in design condition at three points. Besides, the $C_p$ is higher in design condition than other condition. Fig. 8 presents frequency spectrum of pressure fluctuation at three points in inlet passage in three conditions. The pressure fluctuation frequency mainly focus on low frequency area, while high frequency area in off condition. In other words, flow pattern is uniform inside the inlet passage and there is no difference between centre and boundary. The amplitude corresponding to 53.35Hz is higher in large flow rate condition than that in design and low flow rate conditions. Furthermore, 53.35Hz is impeller blade frequency, which means that the flow pattern inside inlet passage is influenced by the flow in impeller.

| Flow rate | Frequency spectrum parameter | P1 | P2 | P3 |
|-----------|-----------------------------|----|----|----|
| $0.667Q_D$ | $f$/Hz                      | 355.7 | 355.7 | 355.7 |
|           | $C_p$                       | 0.00536 | 0.00536 | 0.00536 |
| $Q_D$     | $f$/Hz                      | 35.57 | 35.57 | 35.57 |
|           | $C_p$                       | 0.00864 | 0.00866 | 0.00865 |
| $1.111Q_D$| $f$/Hz                      | 551.3 | 551.3 | 551.3 |
|           | $C_p$                       | 0.007086 | 0.007086 | 0.007086 |
Fig. 10 Frequency spectrum of pressure fluctuation at P1, P2, P3

Table 3 gives frequency spectrum parameters of P4, P5 and P6 in outlet passage. Fig. 9 presents frequency spectrum of pressure fluctuation at P4, P5 and P6. The dominant frequency is 35.57Hz ($2f_r$) at three points in design condition, while the P1-P3’s dominant frequency is also 35.57Hz ($2f_r$) in design condition. In other words, both inlet passage and outlet passage’s dominant frequency is 35.57Hz ($2f_r$) in design condition. Besides, the same result can be get in off design condition.

In off-design condition, the high frequency fluctuation is obviously found in inlet and outlet passage. It means that the flow pattern is worse in off-design condition than that in design condition.

Table 3 Frequency spectrum parameters of P4, P5, P6

| Flow rate   | spectrum | P4     | P5     | P6     |
|-------------|----------|--------|--------|--------|
| $0.667Q_D$  | $f$/Hz   | 355.7  | 355.7  | 355.7  |
|             | $C_p$    | 0.001565 | 0.001549 | 0.001577 |
| $Q_D$       | $f$/Hz   | 35.57  | 35.57  | 35.57  |
|             | $C_p$    | 0.002549 | 0.002495 | 0.002561 |
| $1.111Q_D$  | $f$/Hz   | 551.3  | 551.3  | 551.3  |
|             | $C_p$    | 0.001974 | 0.001952 | 0.001986 |
4. Experimental verification

The model of the diving tubular pump system is tested in high-precise hydraulic machinery test rig. The pump system model is horizontal installed. The pump shaft pass through the outlet passage and connect with torque and rotating speed sensor by shaft coupling. The installed pump system model shows in Fig.12. The uncertainty of efficiency testing system is ±0.39% and is A level based on Chinese Ministry of Water Resources industry standard, SL140-2006.

The results of CFD about energy performance of pump system is compared with that of test. Fig.13 shows the comparison between them. It can be found that the trend of hydraulic performance that is predicted by CFD is in accordance with the testing results. The efficiency predicted by CFD is 72.2%, while the efficiency is 72.9% in test in design condition. The relatively uncertainty of efficiency between CFD and test is 0.96%. The head predicted by CFD is 1.222m, while the head is 1.311m in test in design condition. The relatively uncertainty of head between CFD and test is 6.79%. Those data shows that the CFD result is relatively correct and can predict the hydraulic performance of pump system.

Fig. 11 Frequency spectrum of pressure fluctuation at P4, P5, P6

Fig.12 The model of diving tubular pump system
5. Conclusion

The asymmetry of the longitudinal structure of inlet passage affects the flow pattern on outlet. Especially at small flow rate condition, structural asymmetry will result in the uneven velocity distribution on the outlet of passage inlet. The axial velocity distribution uniformity increases as the flow rate increases on the inlet of passage inlet, and there is a positive correlation between hydraulic loss in the passage inlet and flow rate’s quadratic. The axial velocity distribution uniformity on the outlet of passage inlet is 90% at design flow rate condition. The predicted result shows the same trend with test result, and the range of high efficiency area between predicted result and test result is almost identical. The dominant frequency of pressure pulsation is low frequency in inlet passage at design condition. The dominant frequency is high frequency in inlet passage at small and large flow rate condition. At large flow rate condition, the flow pattern is significantly affected by the rotation of impeller in inlet passage. At off-design condition, the pressure pulsation is strong at outlet passage. At design condition, the dominant frequency is 35.57Hz, which is double rotation frequency.

6. Acknowledgment

This work was financially supported by National sci-tech support plan for 12th five-year plan (2015BAD20B01-02), National science and technology plan project in rural areas (Grant No.2012BAD08B03), National Natural Science Foundation of China (Grant No. 51179167), Water conservancy science and technology project of Jiangsu Province (Grant No.2014046), The Young academic Leaders in Blue Project of Jiangsu, Prospective Joint Research Project of Jiangsu (Grant No. BY2015061-12) , A Project Funded by the Priority Academic Program Development of Jiangsu Higher Education Institutions and Major Natural Science Foundation of Universities in Jiangsu(Grant No.12KJA570001), A Project Funded by the Priority Academic Program Development of Jiangsu Higher Education Institutions,

References

[1] ZHANG D, SHI W, Bin C, et al. Unsteady flow analysis and experimental investigation of axial-flow pump[J]. Journal of Hydrodynamics, Ser. B, 2010, 22(1): 35-43.
[2] Alpan K, Peng W W. Suction reverse flow in an axial-flow pump[J]. Journal of fluids engineering, 1991, 113(1): 90-97.
[3] Xu Hui. Tubular Pump Station[M]. Beijing: China Water & Power Press, 2008 (in Chinese)
[4] Tang Fangping, Liu Chao, Xie Weidong, et al. Experimental studies on hydraulic models for a reversible, tubular, and submersible axial-flow pump installation[J]. Transactions of the Chinese Society for Agricultural Machinery, 2004, 35(5):74－77. (in Chinese)
[5] Cheng Li, Liu Chao, Zhou Jiren. Numerical simulation of flow in the outlet passages of reversible pumping station by RNG turbulent model with wall function law[J]. Advances in Water
Science, 2004, 15(1):109-112. (in Chinese)

[6] Yang Fan, Jin Yan, Liu Chao, et al. Numerical analysis and performance test on diving tubular pumping system with symmetric aerofoil blade[J]. Transactions of the CSAE, 2012, 28(16):60-67. (in Chinese)

[7] Jin Yan, Liu Chao, Tang Fangping, et al. Influence of supporting shape on the hydraulic performance of bulb tubular pumps[J]. Transactions of the Chinese Society for Agricultural Machinery, 2009, 40(11):78-82. (in Chinese)

[8] Liu chao. Researches and Developments of Axial-flow Pump System[J]. Transactions of the Chinese Society for Agricultural Machinery, 2015, 46(6):49-59. (in Chinese)

[9] Zhang N, Yang M G, Gao B, et al. Experimental investigation on unsteady pressure pulsation in a centrifugal pump with special slope volute[J]. Journal of Fluids Engineering, 2015, 137(6): 061103.

[10] Li Y, Wang F. Numerical investigation of performance of an axial-flow pump with inducer[J]. Journal of Hydrodynamics, Ser. B, 2007, 19(6): 705-711.

[11] Zhang N, Yang M G, Gao B, et al. Experimental investigation on unsteady pressure pulsation in a centrifugal pump with special slope volute[J]. Journal of Fluids Engineering, 2015, 137(6): 061103.

[12] The Test on Pump Model in South-to-North Water Diversion Project[M]. China Water & Power Press, 2006 (in Chinese)

[13] Han Z, Reitz R D. Turbulence modeling of internal combustion engines using RNG k-ε models[J]. Combustion science and technology, 1995, 106(4-6): 267-295.