Proceedings of the ASME 2010 3rd Joint US-European Fluids Engineering Summer Meeting and 8th International Conference on Nanochannels, Microchannels, and Minichannels
FEDSM-ICNMM2010
August 1-5, 2010, Montreal, Canada

FEDSM-ICNMM2010-30441

NUMERICAL ANALYSIS OF UNSTEADY FLOW IN CENTRIFUGAL PUMPS WITH IMPELLERS BASED ON DIFFERENT HYDRAULIC DESIGN PRINCIPLES

YUAN Jianping
Research Center of Fluid Machinery Engineering and Technology, Jiangsu University
Zhenjiang, Jiangsu Province, China
Email: yh@ujs.edu.cn

HE Hongqin
Research Center of Fluid Machinery Engineering and Technology, Jiangsu University
Zhenjiang, Jiangsu Province, China
Email: hehq@shxike.com.cn

HE Zhixia
School of Energy and Power Engineering, Jiangsu University
Zhenjiang, Jiangsu Province, China
Email: zxhe@ujs.edu.cn

YUAN Shouqi
Research Center of Fluid Machinery Engineering and Technology, Jiangsu University
Zhenjiang, Jiangsu Province, China
Email: shouqiy@ujs.edu.cn

ABSTRACT

The centrifugal pump is one of the most widely used pumps. Lower efficiency, worse cavitation performance and stronger unsteady flow are three main problems for the low-specific-speed centrifugal pumps. Due to unsteady flow within pumps, pressure is fluctuant, which further induces vibration and noise. For a centrifugal pump with different impellers, its characteristics of unsteady flow are also different. In this paper, five different impellers were designed adopting low-specific-speed design method, splitter-blades design method and velocity-coefficient method with a set of performance parameters (Q=25m^3/h, H=10m, n=1450r/min). 3D unsteady turbulent flow field within the centrifugal pump was simulated. The periodic fluctuation phenomenon and the unsteady flow characteristics were investigated. The static pressure fluctuation in the volute and at the volute outlet and the instantaneous head changes were showed for the pumps with five different impellers operated at design and off-design conditions. The pressure fluctuation is the strongest near the tongue and is slighter at the volute outlet section. The transient head fluctuation increases with the flow rate. The transient head fluctuation of the pump with the low-specific-speed designed impeller is the biggest, while that of the pump with the normal-designed impeller is the smallest. Among the three splitter-blade impellers, the transient head fluctuation of the impeller with splitter blades leaned to the corresponding suction side of the long blades is the smallest. Eventually, the different design schemes were evaluated using the unsteady flow analysis. These conclusions from this paper can supply some references for the design of low-specific-speed centrifugal pumps considering its pressure fluctuation and flow-induced vibration and noise.

INTRODUCTION

Low-specific-speed centrifugal pumps are widely used in nuclear industries, petroleum, aerospace and chemical industries to deliver low flow rate and high-head liquids, but there exist many problems to be solved, such as low efficiency due to disc loss, heavy cavitation erosion and low flow rate instability due to positive slope of head-capacity characteristic curve and unsteady internal flow [1]. For this unsteady internal flow, the fluctuation of the flow velocity, pressure and forces acted on impeller, guide vane and volute may induce the vibration and noise, so it is necessary to investigate the complicated 3D unsteady flow in centrifugal pumps in detail.

Due to high cost and difficulties of visual experiment of the flow in centrifugal pumps, for a long time, many researchers have put focus on numerical simulations and achieve great results. From 1990s, emphasis has been put on the three dimensional viscous steady numerical simulation of the flow in centrifugal pumps. Goede et al. [2] published his numerical research results on 3D viscous flow in centrifugal pumps. Hu et al. [3] carried out numerical studies on 3D viscous turbulent flow in centrifugal pumps. Yuan et al. [4, 5] conducted lots of studies on 3D incompressible turbulent flow calculation and solid-liquid two-phase turbulent flow calculation in non-clogging centrifugal pumps.

In recent years, with the rapid progress of computer technology and computational fluid dynamics (CFD), numerical studies on the unsteady flow in centrifugal pumps have been more and more deeply and widely carried out. In papers [6-8], unsteady flow simulation considering of the interaction between radial diffusers and the centrifugal impeller was performed to analyze the unevenness of the flow in circumferential direction at the impeller outlet and guide vane inlet. In papers [9-12], the frozen rotor approach or sliding grid technique was used for transforming the coupling information between the impeller and volute to finish the 3D turbulent flow numerical simulation of the full passage of the pump, and then revealed that in volute the secondary flow will mainly affect the region located near the impeller outlet. These studies also indicated that numerical simulation of the flow in pumps with consideration of rotor-stator interaction will be more accurate. Jiang et al. [13] simulated the interior flow field by an LES-based CFD program and the vibration of the pump's structure using a parallel explicit dynamic FEM code. In his simulation, a time series of pressure fluctuations on the internal surface as force-boundary conditions was provided and then the vibration of the outer surface of the structure was calculated to eventually clarify the mechanisms of resonant noise generation and propagation. Xu et al. [14] calculated the unsteady turbulent flow in high speed pump caused by rotor-stator interaction using three dimensions Navier-Stokes formula and RNG k-ε model and analyzed the characters of pressure fluctuation induced by the unsteady flow. Fortes-Patella et al. [15] performed 2d unsteady simulation of impeller-volute interaction in centrifugal pumps using overset mesh technology and phase-lag periodic boundary conditions. Gonzalez et al. [16, 17] conducted a numerical simulation and compared the results with the experimental data to capture the dynamic and unsteady flow effects in a centrifugal pump due to the impeller-volute interaction and finally discussed the unsteady pressure fluctuations pattern in the volute shroud. Liu et al. [18] calculated the unsteady flow in double-channel pumps. The pressure changing law of time domain at the outlet of the volute was investigated under different operating conditions and made a conclusion that the pressure fluctuation at the outlet of the volute is closely related to the interaction between impeller and volute. Yamade et al. [19] simulated the unsteady flow in mixed-flow pumps with large eddy simulation (LES) approach . Their study showed that the unsteady simulation based on LES is accepted in a wide range of flow rate.

From the foregoing, the flow in centrifugal pumps is unsteady and the impeller is the key part to greatly affect the overall performance of a centrifugal pump. This unsteady flow in pumps with different impellers is different. Thus, in this paper different impellers were designed based on different hydraulic design principles, such as low-specific-speed design method, splitter-blades design method and velocity-coefficient method, and numerical investigation of unsteady flow in a centrifugal pump with these different impellers were performed to eventually put forward the optimization ideas of low-specific-speed pumps.

NOMENCLATURE

Q	Flow rate, [m^3/h]
H	Head, [m]
n	Rotational speed, [r/min]
n_s	Specific speed, $n_s = 3.65n\sqrt{Q}/H^{3/4}$, [-]
d_h	Hub diameter, [mm]
D_j	Impeller inlet diameter, [mm]
β_2	Blade outlet angle, [°]
D_2	Impeller outlet diameter, [mm]
b_2	Impeller outlet width, [mm]
Z	Number of impeller blades, [-]
φ	Wrap angle of impeller blades，[°]
β_1	Blade inlet angle, [°]
Δt	Length of time step for unsteady simulation, [s]
T	Time, [s]
T	Time cycle, [s]
p	Pressure, [Pa]

GEOMETRICAL MODEL

The main design parameters of a low-specific-speed centrifugal pump studied are Q=25m^3/h, H=10 m, n=1450 r/min, n_s=78, where Q, H, n and n_s are flow rate, head, rotation speed and specific speed at the design point respectively. Based on different hydraulic design principles, five impellers were designed and used for unsteady numerical simulations and analysis.

Case 1

The impeller of case 1 was designed with low-specific-speed design method [1]. The main structure parameters are listed in table 1 and the hydraulic design draw is showed in Fig.1 (a).

Table 1 Main structure parameters of impeller 1

Impeller inlet diameter	D_j	75mm
Hub diameter	d_h	40mm
Impeller outlet diameter	D_2	188mm
Impeller outlet width	b_2	12mm
Number of impeller blades	Z	4
Wrap angle of impeller blades	φ	160°
Blade inlet angle	β_1	23.4
Blade outlet angle	β_2	20°

Case 2

The impeller of case 2 was designed with traditional velocity-coefficient method [20]. The main structure parameters are listed in table 2 and the hydraulic design draw is showed in Fig.1 (b).

Table 2 Main structure parameters of impeller 2

Impeller inlet diameter	D_j	75mm
Hub diameter	d_h	40mm
Impeller outlet diameter	D_2	180mm
Impeller outlet width	b_2	9mm
Number of impeller blades	Z	6
Wrap angle of impeller blades	φ	80°
Blade inlet angle	β_1	29.5
Blade outlet angle	β_2	41

Case 3

The impeller of case 3 was designed with splitter blades. Splitter blades can both avoid jam in the inlet of the impeller for too many blades and the severe divergence of the flow passage in the outlet of the impeller for too few blades. As table 2 listed, the impeller 2 has six blades. We shorted the length of three interval blades to get the impeller 3. The inlet diameter of short blades of impeller 3 is $0.7D_2$ and the short blades are not deviated, showed in hydraulic design draw Fig.1 (c).

Case 4

In order to investigate the influence of deviation of short blades on the internal flow, the impeller of case 4 was designed with a deviation of 10° for short blades rotated to pressure side of long blades, seen in Fig.1 (d).

Case 5

In order to investigate the influence of deviation direction of short blades on the internal flow, the impeller of case 5 was designed with a deviation of 10° for short blades rotated to suction side of long blades, seen in Fig.1 (e).

UNSTEADY NUMERICAL METHODS

In this paper, the unsteady turbulent numerical simulations were finished in Fluent6.3. We have two domains to be simulated, which are composed of an impeller domain and a volute domain. In our study, five impellers were respectively coupled with the same volute to form the computational flow passage. The 3D models for calculation were generated in Pro/E and the computational grids for different impellers were formed by the domain-subdividing and matching method in Gambit, which is the pre-processor of Fluent, shown in Fig.2. A local mesh refinement scheme was used.

For turbulence modeling, a RNG k-ε turbulence closure in the RANS approach has been adopted. The details on the fundamental mathematical models, such as continuity equation, momentum equation and k-ε equations have been given in the reference [21].

We took the impeller inlet section as the inlet boundary for calculation, at which the velocity-inlet boundary condition was applied. The volute outlet was taken as the outlet boundary for calculation and the boundary condition was assumed to be fully developed flow. For the wall, the boundary conditions were defined as the impermeability and no-slip for the velocity. A standard wall-function was also used for the turbulence modeling. In calculation, steady simulation ought to be firstly conducted to supply initial conditions for the unsteady calculations. Due to the impeller's rotation, a sliding interface technique was adopted to simulate the rotor-stator interaction in the absolute coordinate system.

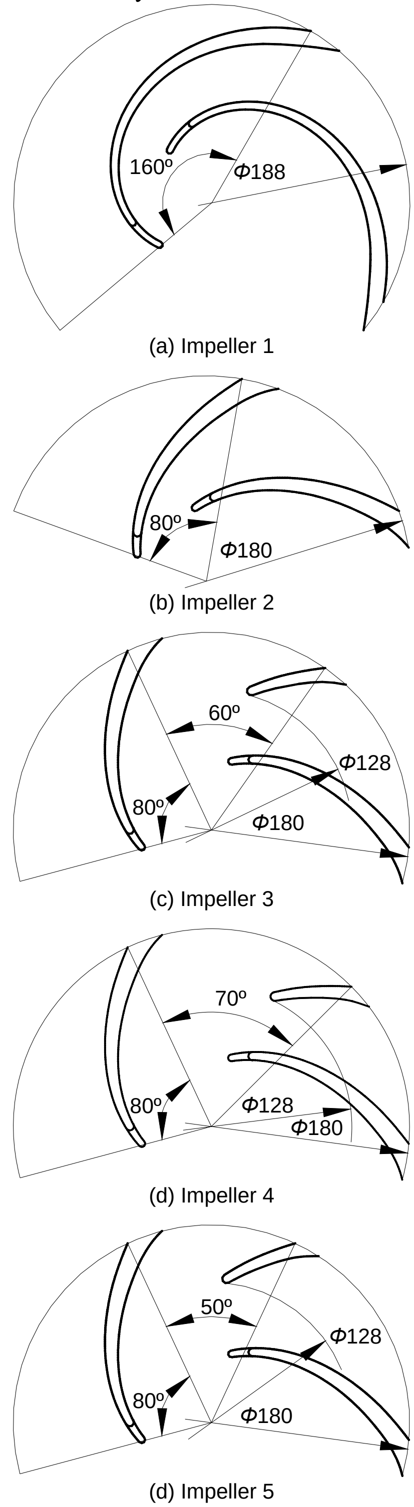


Fig.1 Impeller hydraulic design draws based on different design principles

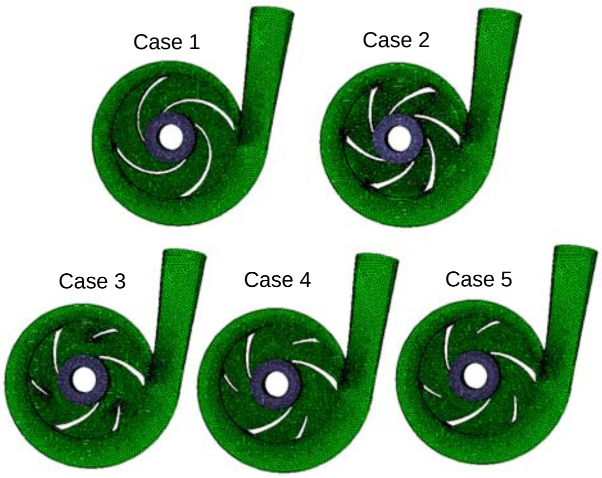


Fig.2 Computational girds for five different cases

The discretization method of these equations was based on the finite volume approach and the coupling of velocity and pressure was achieved using a SIMPLEC algorithm. The convective terms in equations were approximated by the second-order upwind differencing scheme. The PRESTO scheme was used for the pressure gradient item. The temporal terms were discredited using a second-order implicit scheme. The time step in the unsteady calculation was 1.149×10^{-4}s.

RESULTS AND ANALYSES

There are two important concepts to help describe the flow characteristics in centrifugal pumps, which are large flow rate zone and small flow rate zone. Some studies on the flow characteristics of the impeller channel in centrifugal pumps have indicated that the flow rate of the different impeller channel is different. The flow rate of the impeller channels located near volute tongue from -40º to 80º (the positive direction of the first section for the volute is defined as 0º) is larger than the average flow rate of the whole impeller channels, so these channels are called large flow rate zone. The other impeller channels located far from the volute tongue from 80° to 320° are called small flow rate zone, where the flow rate is smaller than the average flow rate.

Unsteady numerical simulations and analyses under design condition

Before comparing and analyzing the flow characteristics in these pumps with different cases, we had firstly chosen the typical case 5 to simulate the flow in whole flow passages including impeller and volute channel and to analyze unsteady flow characteristics in centrifugal pumps and the interaction between its impeller and volute.

Figure 3 and Figure 4 show respectively the static pressure distributions and velocity distributions of case 5 on impeller midsection in a rotating period under design condition. In Fig.3, with the rotation of the impeller, the relative locations of blades to the volute tongue are varied and then the static pressure distribution is also quite different. Especially in large flow rate zone, the variation of static pressure is obvious. The static pressure at inlet of blades may decrease obviously, which is closely related to the relative position between the impeller and volute.

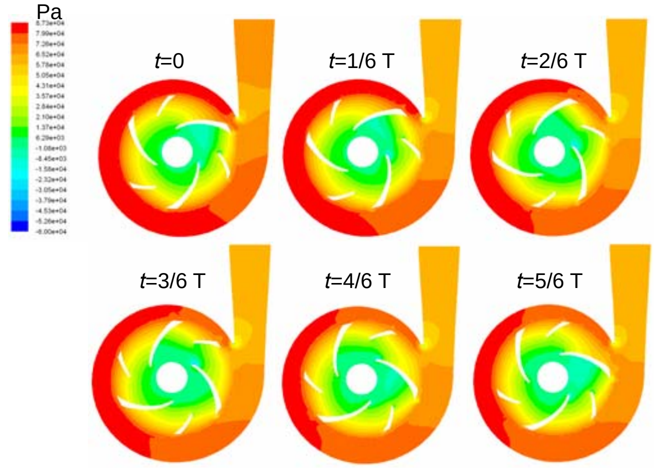


Fig.3 Static pressure distribution on midsection in a rotating period under design condition

In Fig.4, because of the interaction between the impeller and volute, the velocity distributions in different impeller channels are different and the velocity at outlet of the impeller channel near the volute outlet is larger than that of the other channels. The asymmetrical flow field caused by the interaction has extended to the region before the blade inlet. The velocity distributions are different at inlet of the impeller channels, as well as that at outlet of the channels. However, Fig.4 also shows that the velocity distributions in the middle areas of the different impeller channels are similar. These conclusions further verified the above definitions of the large flow rate zone and small flow rate zone. That is, impeller channel zone from -40º to 80º is large flow rate zone, where the velocity is larger and impeller channel zone from 80º to 320º is small flow rate zone, where the velocity is relatively smaller, especially in impeller channel zone from 120° to 200°, where the velocity is much lower closely related with the volute tongue.

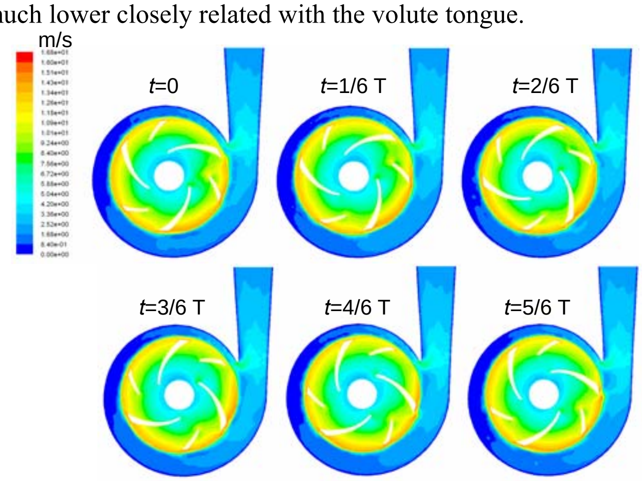


Fig.4 Velocity distributions on midsection in a rotating period under design condition

Unsteady numerical simulations and analyses under different conditions

For low-specific-speed centrifugal pumps, the flow characteristics in pumps under off-design conditions will also be important, as well as that under design condition. So the unsteady numerical simulation for case 5 under off-design conditions had also been carried out, such as low flow rate condition (Q=10m^3/h) and high flow rate condition (Q=40m^3/h). The static pressure distributions on midsection at t=0 and t=1/2T under different flow rate conditions are shown in Fig.5. Under low flow rate condition, the pressure varies a little with the rotation of the impeller at t=0 and t=1/2T. However, under design flow rate condition, the static pressure distribution are quite different in the impeller channel close to the volute tongue. Under high flow rate condition, the pressure distribution varies not only in the impeller channel close to the volute tongue but also in other impeller channels. These simulation results imply that the influence of the interaction between the impeller and volute on the flow field in pumps will be larger and larger with increase of the flow rate.

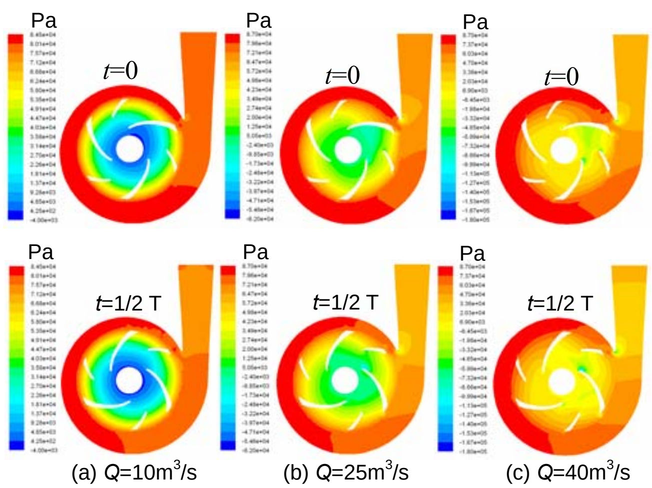


Fig.5 Static pressure distribution in midsection under different conditions at at t=0 and t=1/2T

Pressure fluctuations in the volute with different impellers

In this paper, four representative points were chosen to detect the pressure fluctuations in the volute. Three detecting points are on the circumference of volute inlet with diameter of 192mm and one point is at the center of volute outlet section, as seen in Fig.6. Under design flow rate condition, the static pressure fluctuations at above four detecting points for a centrifugal pump with case 1 in a rotating period are also shown in Fig.6. The impeller of case 1 has four blades, so it will rotate 90° in a period. In Fig.6, the angle of the impeller rotates here is stood for by the variable of φ, which also reflects the time denoted on the horizontal axis. When the end tip of the blade is just opposite to the volute tongue, φ will equal to 0°.

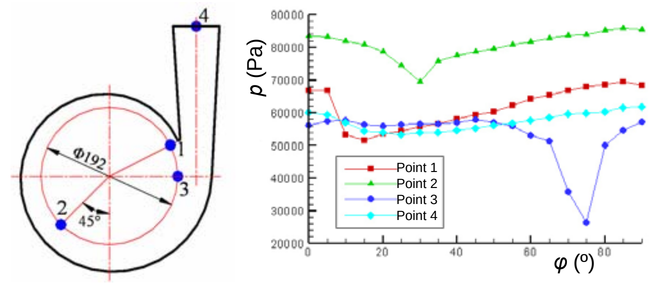


Fig.6 Static pressure at detecting points in a period under design condition for the centrifugal pump with case 1

From the Fig.6, it can be seen that there exists obvious pressure fluctuation in volute due to the interaction between impeller blades and the volute tongue and the fluctuating amplitudes appear different at different locations. The amplitudes at point 1 and point 3 that are both close to the volute tongue are larger than that at point 2 located far from the volute tongue, which indicate a stronger unsteady flow characteristics in areas near the volute tongue. With the rotation of impeller, the time when the pressure fluctuation amplitude is larger at some location depends on the relative position of impeller blades to the volute tongue. When the blade rotationally sweeps the volute tongue for 15°, the pressure at point 1 will drop to the minimum value, while sweeps for 75°, the pressure at point 3 drops to the minimum. Accordingly, the pressure at point 2 drops to the minimum with the value φ of 25°. Thus, with rotation of the impeller, the four blades sweep the volute tongue frequently and then the pressures at above detecting points periodically fluctuate. It can be derived that the periodic pressure fluctuation will appear at any locations in volute, but the fluctuation amplitude will vary at different locations. In addition, there also exists pressure fluctuation at volute outlet section, where the fluctuation amplitude will be quite smaller accordingly.

Figure 7 shows the static pressure fluctuations at above four detecting points under different flow rate conditions. It can be found that the pressure fluctuation amplitudes on all four detecting points are smaller under low flow rate condition and they will be larger and larger obviously with increase of the flow rate. Therefore we conclude that the interaction between the impeller and volute has little effects on the flow characteristics under low flow rate condition. Especially at the volute outlet section, there is nearly no pressure fluctuation.

According to above discussions, the representative detecting point 1 and point 4 chosen, numerical comparisons and analyses of static pressure fluctuations in volutes coupled with five different impellers based on different hydraulic design principles were carried out. The detecting points chosen are positioned on the same axial-sections of volute for five different impellers. As can be seen in Fig.8, the pressure fluctuation amplitude at detecting point 1 for case 1 appears obviously larger than that for other cases, which can be attributed to that the detecting point 1 for case 1 is much closer in radial direction to the blade outlet due to a larger impeller diameter of 188mm compared to the diameter of 180mm of others.

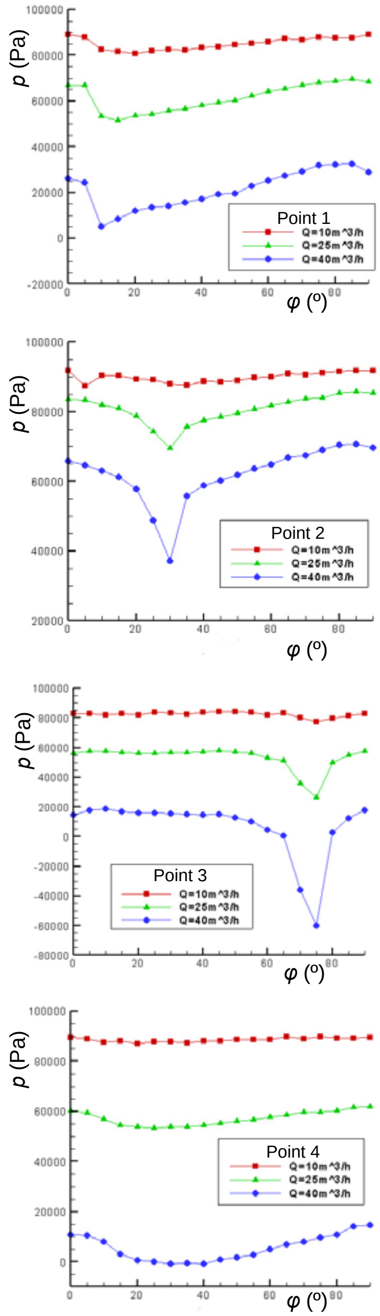


Fig.7 Static pressure at four detecting points in a period under different flow rate conditions for the centrifugal pump with case 1

Moreover, the impeller of case 1 has four blades, which implies the interval duration of every blade rotationally sweeping the volute tongue will be longer compared to the six-blade impeller of the other cases, so the pressure fluctuation has enough time to develop and rise to a larger value. The impellers of case 2 and 3 both have six blades, but the impeller of case 3 was designed with splitter blades, as described in geometrical model that three are short and three are long. Then as can been seen in Fig.8, with rotation of the impeller, during the period of 60° after the short blade of case 3 rotationally sweeping across the volute tongue, the pressure fluctuation at detecting point 1 is similar to that of the case 2, with a normal six-blade impeller, while during the period of 60° before the short blade of case 3 rotationally sweeping across the volute tongue, it seems differently smaller than that of case 2, which results from the better of the impeller inlet flow conditions for the impeller with splitter blades. The impellers of case 3, 4 and 5 were all designed with splitter blades but with different deviation directions of short blades, which can differently affect the flow field in centrifugal pumps. The pressure fluctuation amplitude at detecting point 1 for case 5 with a bias of short blades toward to suction side of long blades is far smaller than that for case 3 and case 4.

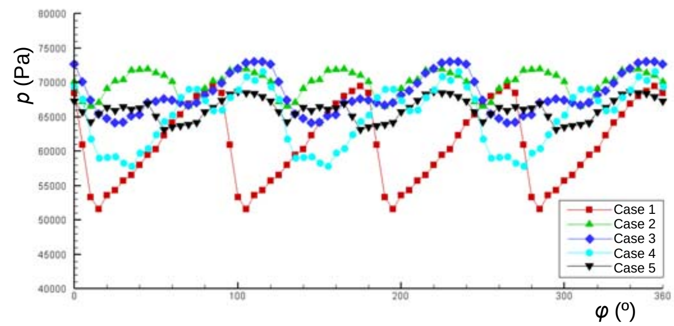


Fig.8 Static pressures at detecting point 1 for five cases in a period under design condition

Figure 9 shows the pressure fluctuations at the detecting point 4 for the above five different cases. Compared with Fig.8, it can been seen that the pressure fluctuation trends at detecting point 4 for all five cases appear similar but the pressure fluctuation amplitudes drop dramatically with the fluid flowing through the whole volute diffuser. Similarly the pressure fluctuation amplitude at detecting point 4 for case 5 is the smallest. Thus, it can be believed that the flow characteristic of case 5 is the best from this aspect.

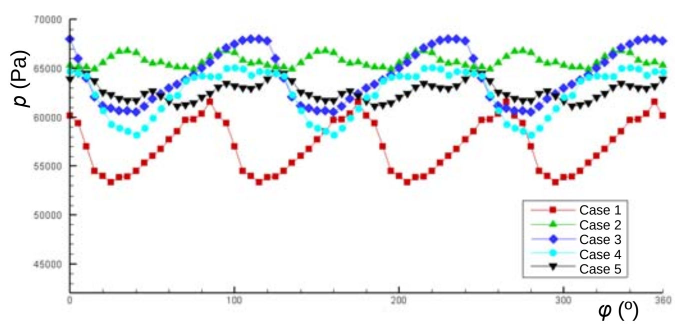
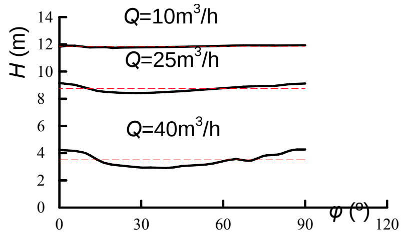


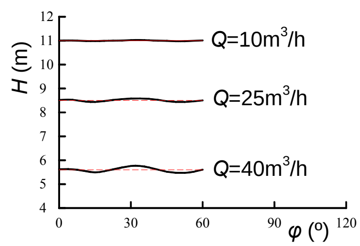
Fig.9 Static pressures at detecting point 4 for five cases in a period under design condition

Instantaneous heads of centrifugal pumps for five cases

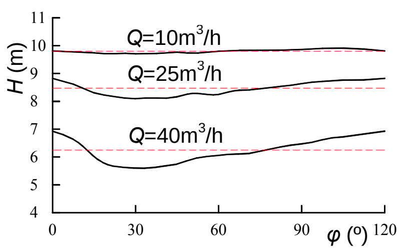
As above analyzed, the interaction between the impeller and volute may induce the unsteady flow in centrifugal pumps and will subsequently result in the variation of instantaneous head. The instantaneous head of centrifugal pumps with five different impellers under different flow rate conditions were shown in Fig.10 and Fig.11.



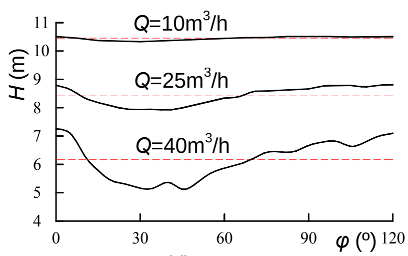
(a) Case 1



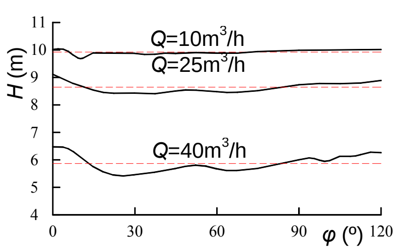
(b) Case 2



(c) Case 3



(d) Case 4



(e) Case 5

Fig.10 Comparisons of instantaneous head of pumps with five impellers under different flow rate conditions

It is believed that the fluid flowing out of the volute is composed of two parts. The first part of fluid flows through the whole volute channel and then out of the volute, so the fluid has enough time to exchange energy among them and mixed thoroughly. The other part of fluid flows directly from the impeller channel close to the volute tongue into the volute diffuser and can't be mixed fully. The second part of fluid will be greatly influenced by the interaction between the impeller's blades and the volute tongue. Thus, with rotation of the impeller, the two parts of fluid with great difference of energy will mix at the outlet region of the volute and then induce the unsteady flow in region near the volute tongue and inside volute diffuser, which can further result in the variation of instantaneous head.

In Fig.10, the red dotted lines denote the average head derived by the instantaneous head of different time. It can be seen that the variation range of the instantaneous head declines with the drops of the flow rate for all five cases, which can be attributed to that the quantity of the fluid directly flowing out from the impeller channel close to the volute tongue will be gradually reduced with the decrease of the flow rate of the pump.

Figure 11 shows that the instantaneous head for five cases all fluctuate gently under low flow rate condition. While the differences of the head fluctuation amplitudes occur under design condition for five different pumps, which is consistent with the difference of the static pressure fluctuation occurring at outlet of volute. For case 5, the instantaneous head fluctuation is steadiest and fluctuation amplitude is the smallest, so it will have better working stability. Then it can be predicted that the flow-induced vibration and noise of the pump of case 5 will also be lower than those of the others. With increase of the flow rate, the head fluctuation amplitude will be higher and higher and also the difference of five pumps will be larger and larger.

CONCLUSIONS

(1) It is verified that there exists strong unsteady flow characteristic because of the interaction between the impeller and volute. Not only the flow structures of different impeller channels were showed different at the same time, but also the flow structure of every impeller channel will change with rotation of the impeller and the periodic fluctuation properties are presented, contributing to the arising of the periodic pressure fluctuation at different circumferential location in the volute.

(2) The flow in volute appears quite complicated. Especially at region close to the volute tongue, the unsteady flow characteristic is most obvious.

(3) The pressure fluctuation existing inside of the impeller and volute has been transferred to the outlet of the volute and then eventually result in the variation of the instantaneous head with the running of centrifugal pumps.

(4) For the centrifugal pump with the impeller designed with a bias of short blades toward to suction side of long blades, the pressure fluctuation amplitudes are smallest both in the whole flow passage of a pump and at outlet of the volute, and the instantaneous head fluctuation is weakest as well.

(5) According to the unsteady numerical analysis, the normal design principles are not feasible for low-specific-speed centrifugal pumps, while the low-specific-speed design method and splitter-blades design method will be better. In terms of the splitter-blades design method, the short blades should tend to the suction side of the long blades.

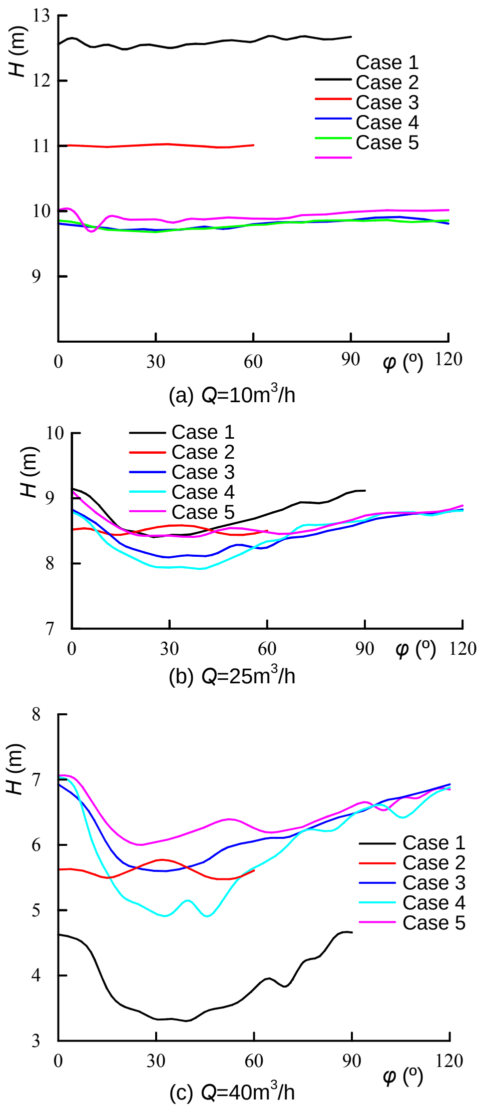


Fig.11 Comparisons of instantaneous heads of pumps with different design principles under three flow rate conditions

ACKNOWLEDGMENTS

This study is supported by National Outstanding Young Scientists Foundation of China (No.50825902), National Natural Science Foundation of China (No.50979034) and Natural science Foundation of Jiangsu Province (No.BK2009218). The authors would like to thank the others of our research team for their working.

REFERENCES

[1] Yuan, S.Q. The theory and design of low specific-speed centrifugal pumps. Beijing: China Machine Press, 1997.

[2] Goede, E., Sebestyen, A., and Schachenmann, A., 1992. "Navier-stokes flow analysis for a pump impeller". In Proceedings of the 16th IAHR Symposium, Sao Paulo, pp. 513-523.

[3] Hu, C.B., Jiang, P.Z., and Wei, J.J., 1999. "Numerical study of turbulent flow on Bingham fluid in pump impeller". *Chinese Journal of Applied Mechanics*, 16(2), pp. 104-107.

[4] Yuan, S.Q., Chen, C., Zheng, M., et al., 2000. "3-d incompressible turbulent flow calculation in the impellers of non-overload". *Chinese Journal of Mechanical Engineering*, 36(5), pp. 31-34.

[5] Yuan, S.Q., Li, Y., and He, Z.H., 2003. "3-d calculation of solid-liquid two-phase turbulent flow within a non-clogging centrifugal pump". *Chinese Journal of Mechanical Engineering*, 39(7), pp. 18-22.

[6] Dawes, W.N., 1995. "A simulation of the unsteady interaction of a centrifugal impeller with its vaned diffuser: flow analysis". *ASME Journal of Turbomachinery*, 117(4), pp. 213-222.

[7] Zhu, B.S., and Kamemoto, K., 2000. "Simulation of unsteady interaction of centrifugal impeller with its diffuser using advanced vortex method". *JSME International Journal*, 43(3), Series B.

[8] Guleren, K.M., and Pinarbasi, A., 2004. "Numerical simulation of the stalled flow within a vaned centrifugal pump". In Proceedings of the Institution of Mechanical Engineers, 218, pp. 425-435.

[9] Guo, P.C., Luo, X.Q., and Liu, S.Z., 2005. "Numerical simulation of 3D turbulent flow fields through a centrifugal pump including impeller and volute casing". *Transactions of The Chinese Society of Agricultural Engineering*, 21(8), pp. 1-5.

[10] Gonzalez, J., Fernandez, J., and Blanco, E., 2002. "Numerical simulation of the dynamic effects due to impeller-volute interaction in a centrifugal pump". *ASME Journal of Fluids Engineering*, 124, pp. 348-355.

[11] Marigorta, E.B., Fernez-Francos, J., and Parrondo-Gayo, J. L., 2000. "Numerical simulation of centrifugal pumps with impeller-volute interaction". In Proceedings of the ASME Fluids Engineering Division Summer Conference, Boston, FEDSM00-11162.

[12] Asuaje, M., Bakir, F., Kouidri, S., et al., 2005. "Numerical modelization of the flow in centrifugal pump: volute influence in velocity and pressure fields". *International Journal of Rotating Machinery*, (3), pp. 244-255.

[13] Jiang, Y.Y., Yoshimura, S., Imai, R., et al., 2007. "Quantitative evaluation of flow-induced structural vibration and noise in turbomachinery by full-scale weakly coupled simulation". *Journal of Fluids and Structures*, 23(4), 531-554.

[14] Xu, Z.H., Wu, Y.L., Chen, N.X., et al., 2004, "Unsteady blade-row interaction calculation in high speed

pump". *Chinese Journal of Mechanical Engineering*, 40(3), pp. 1-4.

[15] Fortes-Patella, R., Longatte, F., and Kueny, J.L., 1995. "Numerical analysis of unsteady flow in a centrifugal pump". *ASME Journal of Fluids Machinery*, FED, (222), pp. 41–46.

[16] González, J., Fernández, J., Blanco, E., et al., 2002. "Numerical simulation of the dynamic effects due to impeller-volute interaction in a centrifugal pump". *ASME Journal of Fluids Engineering*, 124, pp. 348-355.

[17] González, J., and Santolaria, C., 2006. "Unsteady flow structure and global variables in a centrifugal pump". *ASME Journal of Fluids Engineering*, 128, pp. 937-946.

[18] Liu, H.L., Lu, M.Z., Lu, B.B., et al., 2009. "Unsteady flow numerical simulation in a double channel pump and measurements of pressure fluctuation at volute outlet". In Proceedings of the ASME Fluids Engineering Division Summer Conference, Vail, Colorado, FEDSM2009-78065.

[19] Yamad, Y., Kato, C., Shimizu, H., et al., 2009. "Large eddy simulation of internal flow of a mixed-flow pump". In Proceedings of the ASME Fluid Engineering Division Summer Meeting, Vail, Colorado, FEDSM 2009-78416.

[20] Guan, X.F. Design manual of modern pump. Beijing: China Astronautic Publishing House, 1995.

[21] Yuan J.P., 2008. "Inner flow PIV measurement and its unsteady turbulent flow numerical simulation for a centrifugal pump with multi-programs". Ph.D. dissertation, Research Center of Fluid Machinery Engineering and Technology, Jiangsu University, China