Vacuum 159 (2019) 239–246

Contents lists available at ScienceDirect

Vacuum

journal homepage: www.elsevier.com/locate/vacuum

The influence of blade outlet angle on the performance of centrifugal pump with high specific speed

Hongchang Ding*, Zikang Li, Xiaobin Gong, Maoshun Li

School of Mechanical and Electronic Engineer, Shandong University of Science and Technology, Qingdao, 266590, People's Republic of China

ARTICLE INFO

Keywords:
Centrifugal pump
Blade outlet angle
High specific speed
Hydraulic performance

ABSTRACT

The blade outlet angle β_2 is one of the key design parameters of the centrifugal pump, which has an important influence on the internal flow field and performance of centrifugal pump. In this paper, five impeller models with different blade outlet angles (23°, 25°, 27°, 29° and 31°) were built by Solidworks and CFturbo under the premise of other impeller parameters unchanged, and numerical simulation of high specific speed (n_s = 192) pump was carried out in the commercial code ANSYS-CFX. Then the experiment was performed to test the hydraulic performance of centrifugal pump. The results show that the change trend of numerical simulation and experiment is similar, and when the flow rate increases gradually, the hydraulic loss of impeller becomes larger with the increase of blade outlet angle, and the blade outlet angle has obvious influence on the efficiency of centrifugal pump at high flow rate. Besides that, it has small influence on the head, which changes relatively larger at low flow rate than that at high flow rate. And all the research results could provide appropriate reference for designers to select reasonable outlet angle.

1. Introduction

High specific speed centrifugal pump is the type of centrifugal pump with specific speed of 150–300, which has larger flow rate and lower head. And it is widely used in the fields such as agricultural engineering, aerospace engineering, chemical industry etc. With the rapid development of computer technology and computational fluid dynamics (CFD), numerical simulation has become an effective method to analyze the inner flow and performance of centrifugal pump. There are many researches on centrifugal pump performance using CFD method in Refs. [1–5], and the researches have proved the feasibility of numerical simulation using CFD method.

As the performance of centrifugal pump depends on many parameters such as blade number, blade wrap angle, blade thickness, blade outlet angle etc., many researchers have studied the influence of different parameters. Liu et al. [6] found that the head of centrifugal pump grows with the increase of blade number, and there is an optimum value for the efficiency and cavitation characteristics. Tan et al. [7] studied the influence of blade wrap angle on centrifugal pump performance, and concluded that the pump with large wrap angle has wide area for high efficiency and stable operation. Pan et al. [8] analyzed the effects of three variation blade angles on centrifugal pump performance, and found that slip theory can accurately calculate the head of the pump at the design operating point. Bacharoudis et al. [9] designed a laboratory pump that could suit radial impellers with the same diameter, and the experimental results showed that the performance curve become smoother and flatter with the increase of outlet blade angle. Combining numerical simulation with experiment, Shi et al. [10] proved that it was feasible to optimize the performance of deep well centrifugal pump by changing the width of impeller outlet. Yang et al. [11] analyzed the influence of variation of blade thickness and profile on centrifugal pump performance. Zhang et al. [12] promoted a new numerical hydraulic design method for the low specific centrifugal pump impeller, which realized fast design and optimization. Cui et al. [13] analyzed the influence of blade outlet angle on inner flow field and performance of low-specific-speed centrifugal pump. Zhou et al. [14] have studied the effect of different rear shroud radius on hydraulic performance.

In addition, Li et al. [15] studied the unsteady flows caused by the interaction between impeller and volute of a high-speed micro centrifugal pump. Yang et al. [16] studied the effects of blade outlet angle and medium temperature on the crystallization rate. Nishi et al. [17] analyzed the radial thrust of a single blade centrifugal impeller with two different blade outlet angles, and concluded that the larger blade outlet angle had better performance. Shigemitsu et al. [18] studied the effect of blade outlet angle on the performance of turbo pump, and

* Corresponding author.
E-mail addresses: dhchang@sdust.edu.cn (H. Ding), 649339038@qq.com (Z. Li), 528173250@qq.com (X. Gong), 1101947185@qq.com (M. Li).

https://doi.org/10.1016/j.vacuum.2018.10.049
Received 13 August 2018; Received in revised form 19 October 2018; Accepted 19 October 2018
Available online 22 October 2018

proposed a high performance design with simple structure. Zhang et al. [19] concluded that the radial force of the impeller increases gradually with increasing β_2. Dong et al. [20] used boundary element method to identify the effect of blade outlet angle, and concluded that a suitable blade outlet angle of 30° could ensure a better comprehensive performance of the PAT. Lang et al. [21] adopted acoustic-vibro-coupling method to calculate the fluid field of centrifugal pump models with different blade outlet angle. Su et al. [22] studied the numerical and experimental method on multi-stage pump as a turbine system.

In summary, CFD numerical simulation has been proved to be a useful method for analyzing the performance of centrifugal pump, and there are some researches about the pump performance with different blade outlet angles, and a lot of achievements have been obtained. But in the exiting studies, the model is mainly concentrated on the low specific speed pump, and there is little study on high specific speed centrifugal pump. In this paper, the model of high specific speed centrifugal pump is built in Solidworks and CFturbo, and the numerical simulation is done by using commercial code ANSYS-CFX. The internal flow and performance of pump with different blade outlet angles are studied, and experiments are carried out to verify the performance of centrifugal pump. The research results could provide appropriate reference for designers to select reasonable blade outlet angle.

2. Computation model and research method

2.1. Computation model

The physical model studied in this paper is a single-stage single-suction centrifugal pump with a high specific speed of 192, which includes pump outlet, inlet, impeller, volute etc., as shown in Fig. 1. And the inlet of the impeller and the outlet of the volute are properly extended to reduce the influence of the boundary conditions on the internal flow.

The design parameters of centrifugal pump are as follows: flow rate $Q_d = 100\,m^3/h$, head $H = 20\,m$, rotational speed $n = 2900r/min$, and other main geometric parameters are shown in Table 1.

In order to study the effect of blade outlet angle on the performance of the centrifugal pump, five impellers with different blade out angle are designed with other parameters unchanged, as shown in Fig. 2. And the shape of impeller blade is designed to be a twisted blade, as the centrifugal pump has a high specific speed.

Table 1
Main parameters of centrifugal pump.

95	143	24	23
Suction diameter D_j/mm	Impeller diameter D_2/mm	Outlet width b_2/mm	Inlet blade angle β_1(°)
Outlet blade angle β_2/(°)	Blade wrap angle Φ/(°)	Blade number Z	Blade thickness δ/mm
27	120	5	3

Fig. 2(a) shows the blade outlet angle in the 2-D sketch of impeller, which is an included angle between tangent and circumferential direction at the edge of blade outlet. Fig. 2(b) shows the 3-D model of impellers established by Solidworks and CFturbo, which are used to perform the numerical simulation in ANSYS-CFX, and there is almost no difference in appearance of the 3-D models with different blade outlet angle.

2.2. Mesh and check of grid independence

The computational domain consists of three parts, namely inlet pipe, impeller and volute, and the grids are generated by using ICEM CFD software. Because of the complex flow passage structure of centrifugal pump, unstructured tetrahedral mesh with good adaptability has been adopted, and local mesh refinement is used to deal with the parts having large flow gradient change. And Fig. 3 shows the generated numerical grids of the whole computational domain, the refined grids of the blade and the tongue of volute.

With the increase of grid numbers, the calculation time will be increased, while the calculation result will tend to be a relative constant value, which is namely the grid independence. In the numerical simulation, three different mesh schemes are chosen to verify the grid independence, as shown in Table 2.

Models of centrifugal pump with three different grid numbers are simulated at design flow rate, and the pump heads are obtained, as

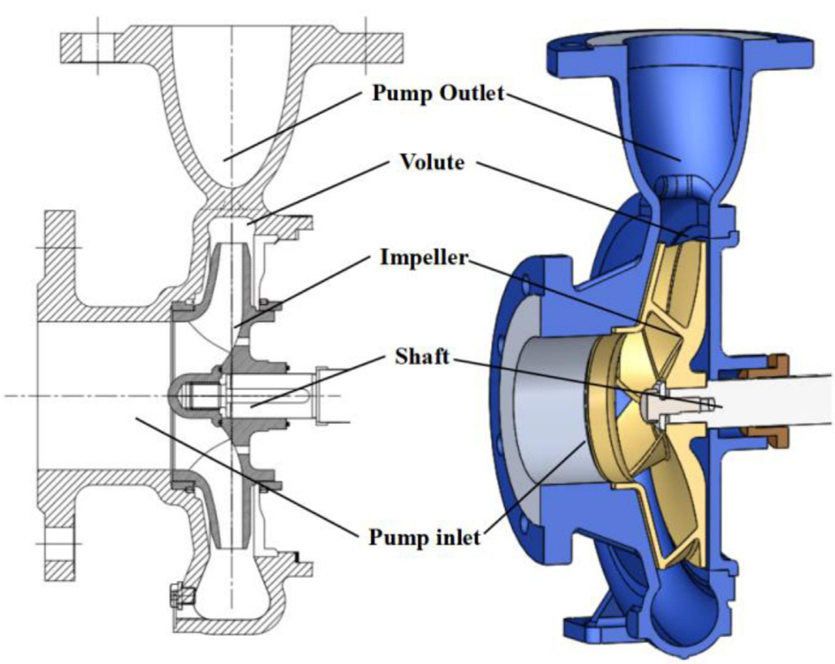

Fig. 1. Physical model of centrifugal pump with high specific speed.

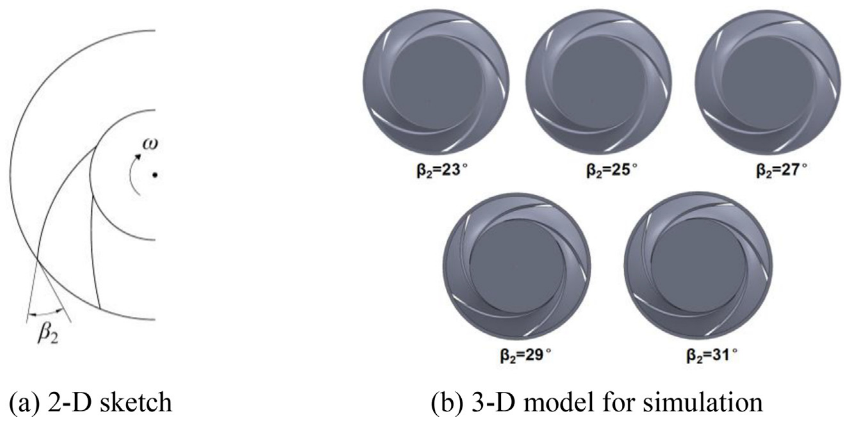

Fig. 2. Impellers with different blade outlet angle β_2.

shown in Fig. 4. It can be seen that the head of scheme I is biggest among the three schemes, and the head of scheme II almost equals that of scheme III, which means that the grid numbers of model increase to a certain value, the predicted head eventually will tend to be a constant value. So the scheme II is finally chosen according to the calculation time and grid independence.

2.3. Governing equation

The RNG k-ε turbulence model is adopted to close Reynolds-averaged Navier-stokes equations [23], and the simulation is carried out by solving the Navier-Stokes equation coupled with a Reynolds average equation. And the conservation equation is expressed as:

$$\frac{\partial U}{\partial t} + \nabla \cdot F_1 + \nabla \cdot F_V = \phi \tag{1}$$

where F_1 and F_V are the non-viscous flux and viscous flux, respectively, U is the variables used for solving the equation, ϕ is a source item.

The turbulent model is a k-ε model, and it can be expressed as:

$$\frac{\partial(\rho u_j k)}{\partial x_j} = \frac{\partial}{\partial x_j}\left(\mu_t \frac{\partial k}{\partial x_j}\right) + G_k - \rho\varepsilon \tag{2}$$

$$\frac{\partial(\rho u_j \varepsilon)}{\partial x_j} = \frac{\partial}{\partial x_j}\left(\frac{\mu_t}{1.3}\frac{\partial k}{\partial x_j}\right) + \frac{\varepsilon}{k}(1.44G_k - 1.92\rho\varepsilon) \tag{3}$$

$$G_k = \mu_t\left({\partial u_i}/{\partial x_j} + {\partial u_j}/{\partial x_i}\right){\partial u_i}/{\partial x_j} \tag{4}$$

where μ_t is the molecular viscosity coefficient, ρ is the density, k is the turbulence energy, G_k is the turbulent kinetic energy caused by velocity gradient, u is velocity of liquid, and ε is the turbulence dissipation rating.

Table 2
Mesh schemes for grid independence check.

Mesh scheme	Inlet	Impeller	Volute
I	308714	210238	493180
II	421208	282283	664958
III	600898	376650	944017

2.4. Research method and simulation conditions

The research method adopts numerical simulation, which is carried out by commercial software ANSYS-CFX. And the numerical simulation is based on the following conditions:

- Fluid in the centrifugal pump is water with a density of 1000 kg/m^3, and the environment temperature is 25 °C.
- The inlet and volute are set to be stationary zone, and the impeller is set to be a rotating zone.
- An isothermal model is chosen as the heat transfer model.
- The inlet boundary condition is set to equal total pressure and the relative pressure is 50000P$_a$. The outlet boundary condition is set to be a mass flow rate.
- For wall boundary condition, no slip is applied on the wall surface, and standard wall function is applied to the adjacent region of the walls.
- The pressure-velocity coupling equation is solved through the SIMPLE algorithm, and the convergence precision of all residuals is less than 10^{-5}.

2.5. Model verification

In order to verify the feasibility of the model pump for simulation, hydraulic performance test is performed on the IS100-80-125 pump,

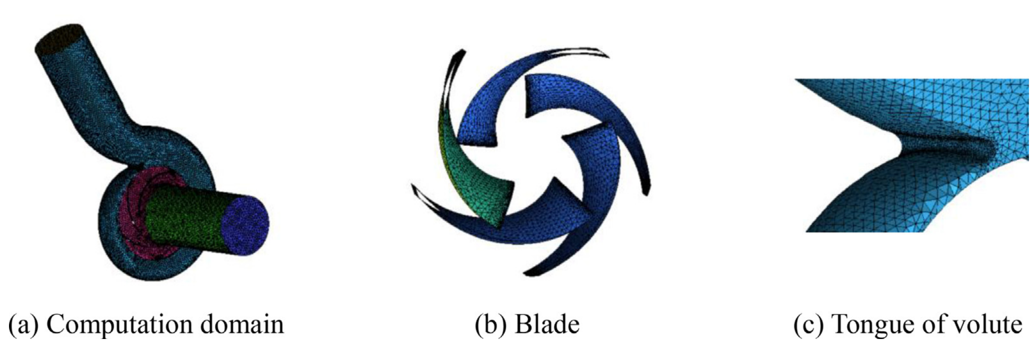

Fig. 3. Numerical grids.

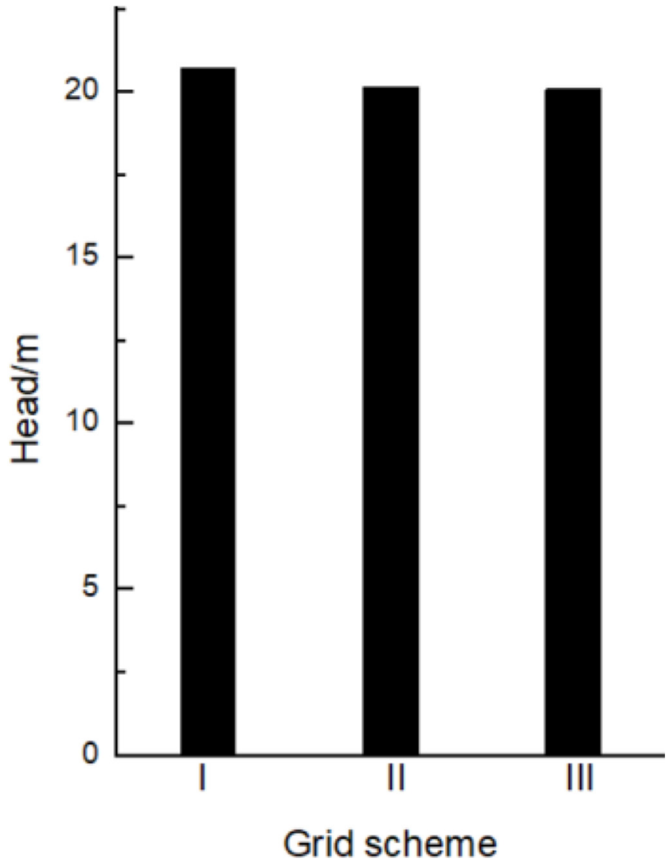

Fig. 4. Grid independence verification.

and the experimental test system is shown in Fig. 5.

The performance test is carried out under five different flow rates, and the *Q-H* curve of experiment is drawn according to the test results. Meanwhile, *Q-H* curve obtained by numerical simulation is also drawn, as shown in Fig. 6. It can be seen that the maximum error between the experiment and simulation is less than 5%, so the model can be simulated.

3. Results and analysis

3.1. Prediction algorithm

Head of centrifugal pump (*H*) is calculated as follows:

$$H = \frac{p_{out} - p_{in}}{\rho g} \tag{5}$$

where p_{out} is the total pressure of volute outlet, p_{in} is the total pressure of impeller inlet, ρ is the density of the fluid, and *g* is the gravity acceleration.

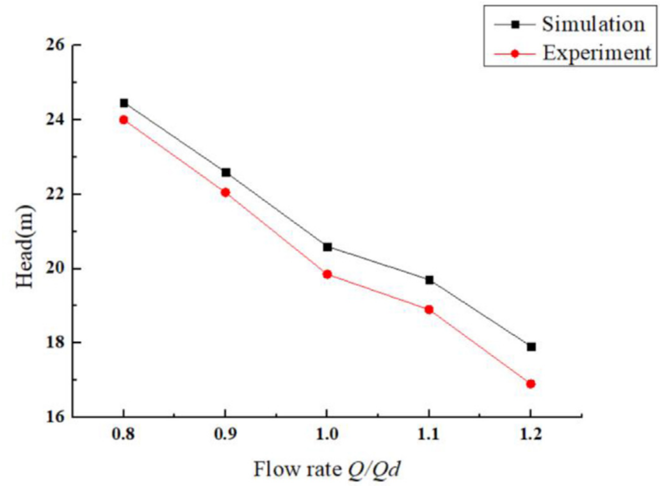

Fig. 6. *Q–H* curves of experiment and simulation for $\beta_2 = 27°$.

Hydraulic efficiency (η_h) is calculated as follows:

$$\eta_h = \frac{\rho g Q H}{M\omega} \tag{6}$$

where *M* is the impeller torque, *ω* is the angle velocity of centrifugal pump.

The volume efficiency (η_v) is expressed as:

$$\eta_v = \frac{1}{1 + 0.68 n_s^{-2/3}} \tag{7}$$

And the total efficiency (*η*) is obtained as follows:

$$\eta = \left(\frac{1}{\eta_v \eta_h} + \frac{\Delta p_d}{p_e} + 0.03\right)^{-1} \tag{8}$$

$$p_e = \rho g Q H \tag{9}$$

where p_e is the effective power of fluid, Δp_d is the disk friction loss, and its calculation method is described in Ref. [24].

3.2. Hydraulic performances of numerical simulation

Supposing that the working flow rate of the centrifugal pump is from 80 m^3/h to 120 m^3/h, and the design flow rate equals 100 m^3/h. According to Eqs. (5)–(9), the external characteristic curves of the centrifugal pump with different blade outlet angles can be obtained, as shown in Fig. 7.

Fig. 7 (a) shows that the head of the centrifugal pump increases with the increase of blade outlet angle at low flow rate of 80 m^3/h. As the flow rate increases, the influence of blade outlet angle on the head becomes weak. Fig. 7 (b) shows that the general trend of efficiency firstly increases and then decreases with the increase of flow rate, and the maximum efficiency corresponds to the design flow rate. At low

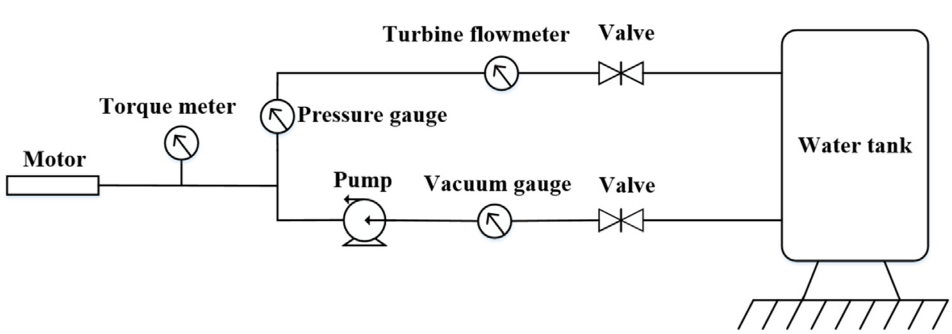

Fig. 5. Experiment test system for hydraulic performance.

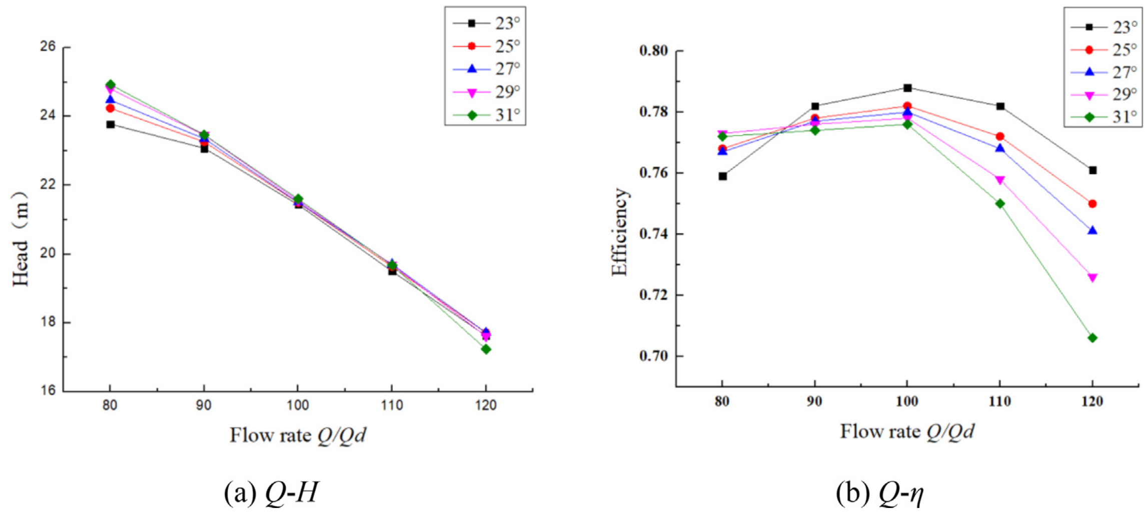

Fig. 7. Hydraulic performances of numerical simulation.

flow rate, the small blade outlet angle has a lower efficiency than that of big angle, and the trend becomes opposite at high flow rate. The maximum efficiency difference between different blade outlet angles is about 1.5% at design flow rate, and when the flow rate increases to $120\ m^3/h$, the efficiency difference reaches to about 5.7%. And it can be concluded that the blade outlet angle has a significant effect on the efficiency of centrifugal pump.

3.3. Pressure distributions

The pressure distributions of impeller are shown in Fig. 8. It can be seen that the pressure gradually increases from the impeller inlet to outlet, the pressure at the inlet has the smallest value and a large gradient. The pressure is bigger on the pressure surface than that on the suction surface at the same position, so the back surface of inlet is the position where cavitation is prone to occurrence. Due to the obstruction effect, the pressure of the tongue part has certain fluctuations. By comparing the pressure distributions in Fig. 8, it can be concluded that the pressure of impeller inlet first decreases and then increases, and the change trend of pressure distribution is consistent under different outlet angles. Note that the pressure near the tongue will increase with the increase of the blade outlet angle at low flow rate condition.

3.4. Relative velocity distribution

Fig. 9 shows the relative velocity distribution of the impeller and volute. It can be seen that the relative velocity of the fluid in the impeller increases gradually, and the velocity has minimum value and smaller gradient in uniform flow at the impeller inlet. At the impeller outlet, with the rise of blade outlet angle, the flow velocity increases and the flow becomes unstable, and the velocity at the pressure surface is bigger than that on the back. When the fluid flows into the volute, the velocity would decrease gradually because of the diffusion structure of volute, and the velocity field of the impeller near the tongue shows certain fluctuations. In addition, the velocity field distribution of the impeller with different outlet angles is consistent, but the velocity field distribution of tongue is quite different.

To observe the relative velocity distributions of volute tongue more clearly, enlarging the volute tongue part of Figs. 9, and Fig.10 can be obtained. In Fig. 10, the velocity gradient at tongue increases with the increase of outlet angle, and the channel between the adjacent blades becomes shorter, which leads to more serious diffusion of velocity in

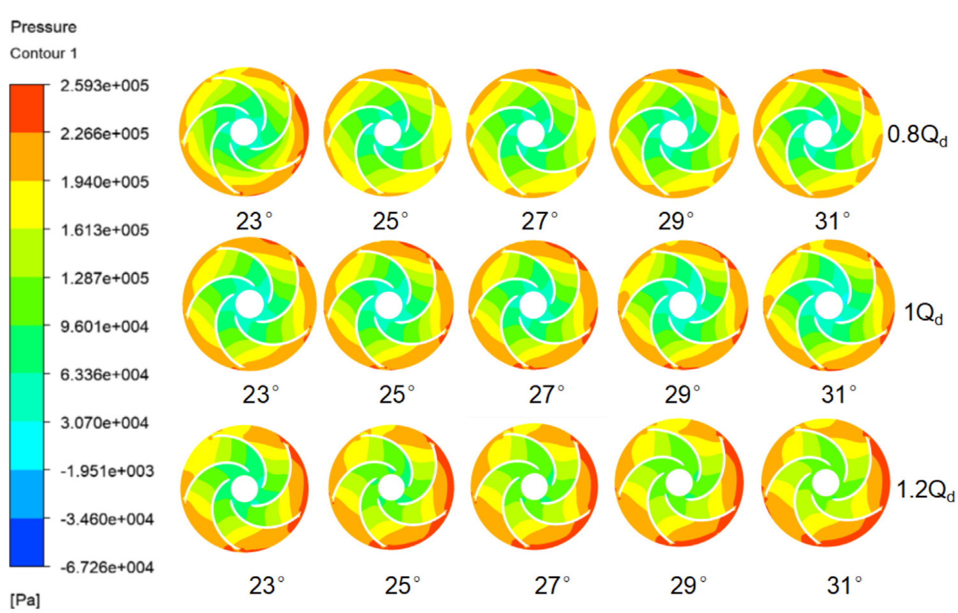

Fig. 8. Pressure distributions of impeller.

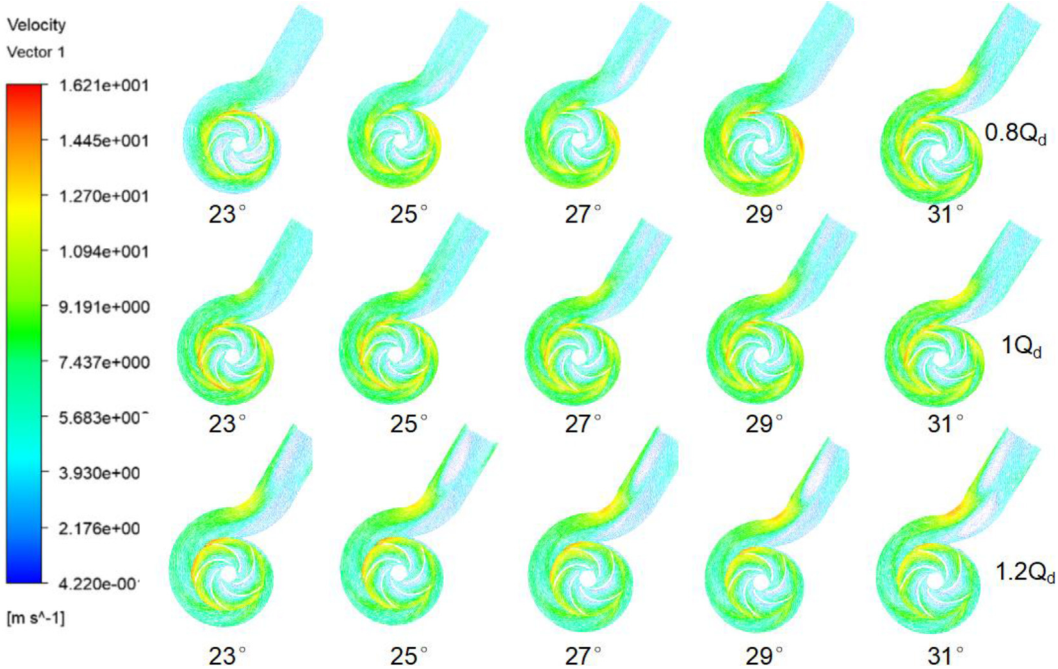

Fig. 9. Relative velocity distributions at midspan of impeller and volute.

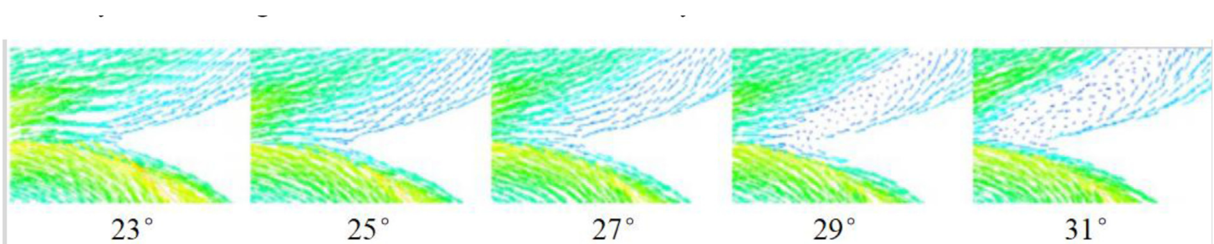

Fig. 10. Relative velocity distributions of volute tongue.

(a) Impellers (b) Test system

Fig. 11. Experimental test platform.

the tongue, and it will cause certain hydraulic loss.

4. Experiment

Fig. 11(a) shows the five rapid prototyping plastic impellers, which are installed to the experimental pump for performance testing. And the external performance of centrifugal pump with different blade outlet angles is tested in the open test rig, as shown in Fig. 11(b). The test rig is composed of three parts: centrifugal pump, water tank and circulation pipeline. The flow rate is regulated by the discharge valve, and the flow rate is measured by the LWGY-MIK turbine flowmeter, whose measuring accuracy reaches to ± 1%R. The centrifugal pump is driven by Y160M1-2 motor, whose parameters are: rated voltage 380 V, rated power 11kw, frequency 50 Hz. And the pressure gauge and vacuum meter are installed at the inlet and outlet of centrifugal pump, which are used to measure the pressure.

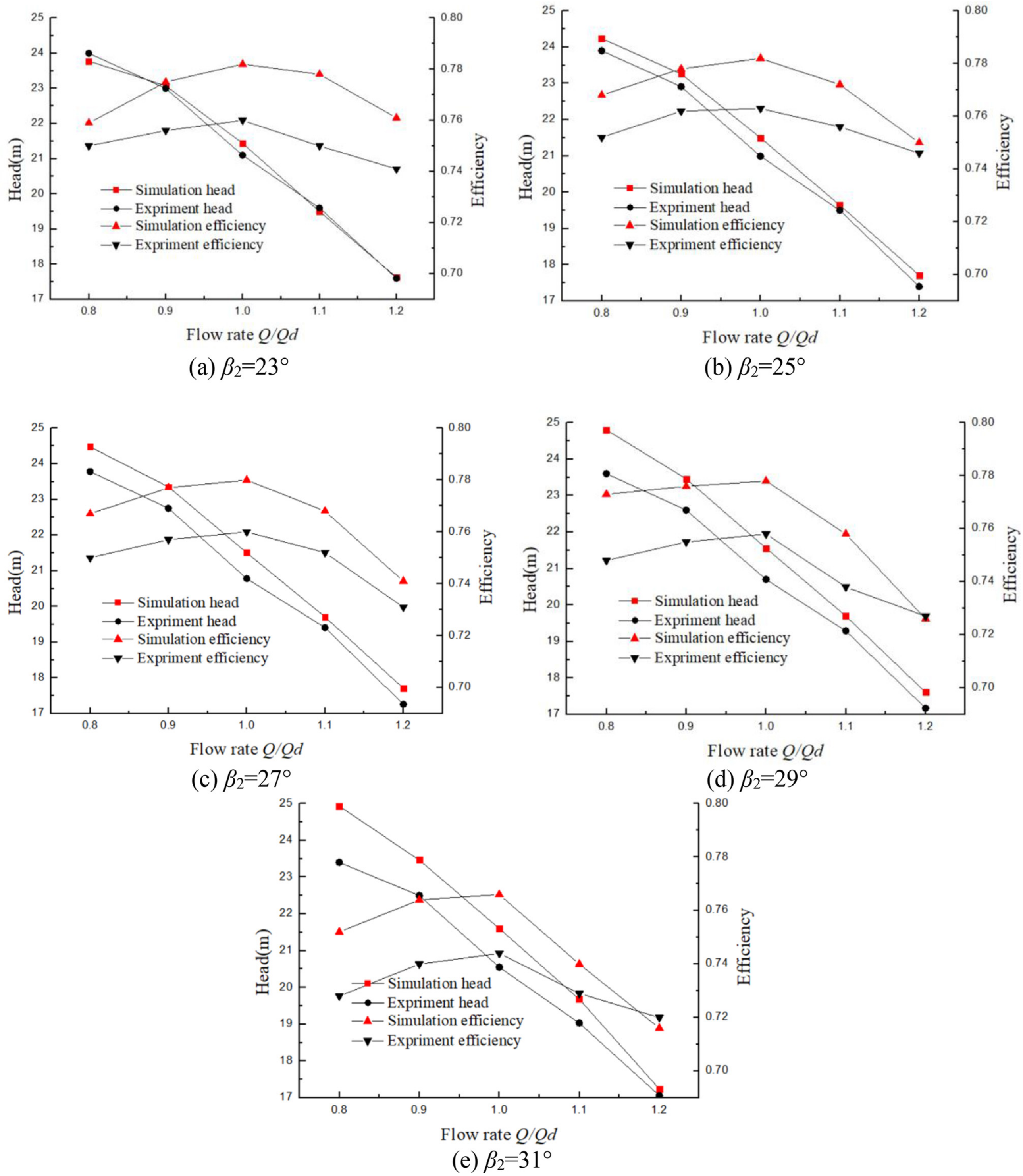

Fig. 12. Comparisons between experiment and calculation curves.

The comparisons between experiment and calculation curves of centrifugal pump with different outlet angle are shown in Fig. 12. It can be seen from Fig. 12 that the head and efficiency of the centrifugal pump decreases with the rise of blade outlet angle at the same rate. At design point, when the blade outlet angle equals 23°, the head and efficiency of experiment are 21.20 m, 76.12%, respectively. When the blade outlet angle equals 31°, the head and efficiency are 20.55 m, 73.94%, respectively. Therefore, it can be concluded from the experimental data that when the blade outlet angle increases from 23° to 31°, the head decreases by 0.65 m, and the efficiency decreases by 2.18%. Therefore, it is necessary to choose the appropriate blade outlet angle when designing the centrifugal pump impeller.

In Fig. 12, it can be seen that the change trend of numerical simulation and experiment is similar, and the results of experiment are less than that of simulation, which are caused by the greater hydraulic loss in experiment. And there is a little difference at low flow rate for the head and efficiency. The max error of head reaches about to 1.8 m between the simulation and experiment at low flow rate with angle of 31°. In addition, the max error of efficiency reaches about 0.03 between simulation and experiment at design flow rate with angle of 31°, and the efficiency of experiment is always less than those of simulation.

5. Conclusions

In this paper, the influences of blade outlet angle on inner flow and characteristics of centrifugal pump are researched by using the methods of numerical simulation and experiment, and the main research conclusions are obtained as follows:

(i) The blade outlet angle has more obvious influence on the efficiency of centrifugal pump, and it has a little influence on the head.
(ii) The pressure at the inlet has small value and large gradient, and the pressure near the tongue increases with the increase of the blade outlet angle at low flow rate.
(iii) The velocity field of the impeller near the tongue has a certain fluctuations, and the velocity gradient increases with the increase of outlet angle at high flow rate, which will enlarge the hydraulic loss.
(iv) The change trend of numerical simulation and experiment is very similar, and the max error of head and efficiency between the simulation and experiment reach about 1.8 m and 0.03, respectively, which responds to the low flow rate with angle of 31°.

Conflicts of interest

The authors declare that there is no conflicts of interests regarding the publication of this paper.

Acknowledgment

This research has been supported by China Postdoctoral Science Foundation (2015M582112), China; key research and development project of Shandong province (2017GGX203005), China; the application and research project of postdoctoral researchers of Qingdao (2014), China. The supports are gratefully acknowledged.

References

[1] S.K. Sun, B. Zhao, X.H. Jia, X.Y. Peng, Three-dimensional numerical simulation and experimental validation of flows in working chambers and inlet/outlet pockets of Roots pump, Vacuum 137 (2017) 195–204.
[2] R.B. Medvitz, R.F. Kunz, D.A. Boger, J.W. Lindau, A.M. Yocum, L.L. Pauley, Performance analysis of cavitating flow in centrifugal pumps using multiphase CFD, J. Fluid Eng. 124 (2002) 377–383.
[3] F.J. Kelecy, Numerical prediction of cavitation in a centrifugal pump, ONETCFD Network Newsletter 3 (2003) 14–16.
[4] W.U. Dazhuan, Y. Shuai, X.U. Binjie, Q. Liu, W.U. Peng, L. Wang, Investigation of CFD calculation method of a centrifugal pump with unshrouded impeller, Chin. J. Mech. Eng. 27 (2014) 376–384.
[5] F.-J. Wang, Y.-J. Li, G.-H. Cong, W.-E. Wang, H.-S. Wang, CFD simulation of 3 D flow in large-bore axial-flow pump with half-elbow suction sump, Journal of Hydrodynamics, Series B 18 (2006) 243–247.
[6] Houlin Liu, Yong Wang, Shouqi Yuan, Minggao Tan, Kai Wang, Effects of blade number on characteristics of centrifugal pumps, Chin. J. Mech. Eng. 23 (2010) 742–747.
[7] T. Lei, B. Zhu, S. Cao, B. Hao, Y. Wang, Influence of blade wrap angle on centrifugal pump performance by numerical and experimental study, Chin. J. Mech. Eng. 27 (2014) 171–177.
[8] P. Zhongyong, X. Rong, C. Yingjie, L. Xiaojun, Analysis effects of blade angle on centrifugal pump performance, Drain. Irrig. Mach. 5 (2009).
[9] E.C. Bacharoudis, A.E. Filios, M.D. Mentzos, D.P. Margaris, Parametric study of a centrifugal pump impeller by varying the outlet blade angle, Open Mech. Eng. J. 2 (2008) 75–83.
[10] W. Shi, Z. Ling, L.U. Weigang, P. Bing, L. Tao, Numerical prediction and performance experiment in a deep-well centrifugal pump with different impeller outlet width, Chin. J. Mech. Eng. 26 (2013) 46–52.
[11] J.H. Yang, J.H. Zhang, Q.C. Sun, X.H. Wang, Influence of variation of blade thickness and profile on centrifugal pump performance, J. Lanzhou Univ. Technol. 37 (2011) 50–55.
[12] Z. Yongxue, Z. Xin, J. Zhongli, J. Cuiwei, Numerical design and performance prediction of low specific speed centrifugal pump impeller, International Journal of Fluid Machinery and Systems 4 (2011) 133–139.
[13] B. Cui, C. Wang, Z. Zhu, Y. Jin, Influence of blade outlet angle on performance of low-specific-speed centrifugal pump, J. Therm. Sci. 22 (2013) 117–122.
[14] L. Zhou, W. Shi, W. Li, R. Agarwal, Numerical and experimental study of axial force and hydraulic performance in a deep-well centrifugal pump with different impeller rear shroud radius, J. Fluid Eng. 135 (2013) 104501.
[15] Y. Li, S. Yuan, H. Lai, Numerical study of unsteady flows with cavitation in a high-speed micro centrifugal pump, J. Therm. Sci. 26 (2017) 18–24.
[16] Minguan Yang, Dong Kang, Dong Xiang, Influence of blade outlet angle on inner flow field of centrifugal pump transporting salt aqueous solution, Chin. J. Mech. Eng. 22 (2009) 912–917.
[17] Y. Nishi, J. Fukutomi, R. Fujiwara, Effect of Blade Outlet Angle on Radial Thrust of Single-blade Centrifugal Pump, (2012), p. 2039.
[18] T. Shigemitsu, J. Fukutomi, R. Nasada, K. Kaji, The effect of blade outlet angle on performance and internal flow condition of mini turbo-pump, J. Therm. Sci. 20 (2011) 32–38.
[19] Y.N. Zhang, W.D. Cao, L.J. Yao, X. Jiang, Analysis on Pressure Fluctuation and Radial Thrust of Centrifugal Pump under Different Blade Outlet Angle, Fluid Machinery, 2017.
[20] L. Dong, C. Dai, F. Kong, L. Fu, Y. Bai, Impact of blade outlet angle on acoustic of centrifugal pump as turbine, Trans. Chin. Soc. Agric. Eng. 31 (2015) 69–75.
[21] D.P. Lang, A.L. Yang, X.U. Yang, D. Ren, The influence of blade outlet angle of a centrifugal pump on the flow induced noise, J. Eng. Thermophys. 35 (2014) 66–69.
[22] X. Su, S. Huang, Y. Li, Z. Zhu, Z. Li, Numerical and experimental research on multi-stage pump as turbine system, Int. J. Green Energy 14 (2017) 996–1004.
[23] V. Yakhot, S.A. Orszag, Renormalization group analysis of turbulence. I. Basic theory, J. Sci. Comput. 57 (1986) 1722.
[24] A.K. Singhal, M.M. Athavale, H. Li, Y. Jiang, Mathematical basis and validation of the full cavitation model, J. Fluid Eng. 124 (2002) 617–624.