DOI: 10.3901/CJME.2013.01.046, available online at www.springerlink.com; www.cjmenet.com; www.cjmenet.com.cn

Numerical Prediction and Performance Experiment in a Deep-well Centrifugal Pump with Different Impeller Outlet Width

SHI Weidong, ZHOU Ling*, LU Weigang, PEI Bing, and LANG Tao

National Research Center of Pumps and Pumping System Engineering and Technology, Jiangsu University, Zhenjiang 212013, China

Received November 1, 2011; revised May 17, 2012; accepted October, 2012

Abstract: The existing research of the deep-well centrifugal pump mainly focuses on reduce the manufacturing cost and improve the pump performance, and how to combine above two aspects together is the most difficult and important topic. In this study, the performances of the deep-well centrifugal pump with four different impeller outlet widths are studied by the numerical, theoretical and experimental methods in this paper. Two stages deep-well centrifugal pump equipped with different impellers are simulated employing the commercial CFD software to solve the Navier-Stokes equations for three-dimensional incompressible steady flow. The sensitivity analyses of the grid size and turbulence model have been performed to improve numerical accuracy. The flow field distributions are acquired and compared under the design operating conditions, including the static pressure, turbulence kinetic energy and velocity. The prototype is manufactured and tested to certify the numerical predicted performance. The numerical results of pump performance are higher than the test results, but their change trends have an acceptable agreement with each other. The performance results indicted that the oversize impeller outlet width leads to poor pump performances and increasing shaft power. Changing the performance of deep-well centrifugal pump by alter impeller outlet width is practicable and convenient, which is worth popularizing in the engineering application. The proposed research enhances the theoretical basis of pump design to improve the performance and reduce the manufacturing cost of deep-well centrifugal pump.

Key words: deep-well centrifugal pump, impeller outlet width, performance, numerical simulation

1 Introduction

Deep-well centrifugal pumps，the main equipment for pumping underground water， have been widely used in the field irrigation, oil extraction, and geothermal utilization and so on[1]. In order to reduce the cost and shorten the production cycle, more and more producers began using plastic injection molding process processing flow components (i.e., impellers and diffusers). In the impeller mold processing process, the impeller blades and rear shroud is one body, and the front shroud is manufactured in another mold. Adjusting the distance of front shroud and rear shroud, namely the impeller outlet width, is the most economical way to change pump hydraulic performance because that does not need change the molds.

The purpose of changing the impeller outlet width is similar with the intention of impeller trimming. Changing the pump performance by impeller trimming is also a point of interest of many authors[2–4]. Impeller trimming is the process of decreasing the impeller diameter to decrease the energy of the system fluid. Impeller trimming provides a useful correction for oversized pumps in application. Trimming an impeller is an alternative to purchasing a smaller impeller from the pump manufacturer. Oftentimes, the next smaller size impeller is too small for the pump load. Moreover, smaller impellers may not be available and impeller trimming is the only practical solution without replacing the entire pump/motor assembly. In summary, it is vital to find out the appropriate impeller trimming method and study its influence on pump performance.

Impeller outlet width is one of the main impeller geometry parameters, which has a significant effect on the performance of rotating machinery. TAN, et al[5], carried out the numerical and experimental study on a centrifugal pump, with six different impeller outlet widths by change the front streamline. The results show that the impeller outlet width has an important effect on the location and area of low pressure region in the blade inlet, jet-wake structure and the back flow in impellers. Three impellers with different outlet widths in centrifugal compressors were studied by REDDY, et al[6]. The results indicated that impeller outlet flow parameters have marked influence on compressor performance, and there is apparently an optimum width for the impeller where the best performance is obtainable. MAHDI, et al[7], described experimental and

* Corresponding author. E-mail: lingzhoo@hotmail.com

This project is supported by National Natural Science Foundation of China (Grant Nos. 51279069, 51109093), and Jiangsu Provincial Natural Science Foundation of China (Grant Nos. BK2011503, BK2011505)

computational analyses of the impeller area ratio by cutting the impeller outlet width in a centrifugal compressor. However, the relate report on centrifugal pump with changing impeller outlet width is extremely rare, and most especially changing the outlet width by offsetting the front shroud.

With the rapid development of computer technology, numerical simulation has been widely used for flow flied analysis and performance prediction of rotating machinery. The precision of numerical simulation has been proved backforth by numerous researchers[8–11].

The present work aims at the study of the pump performance with four different impeller outlet widths. All the four studied impellers have the same blade geometry and fitting with the same diffuser separately. Numerical simulations were carried out in Fluent to analyze the pump performance and flow field distribution. In order to study the accuracy of numerical simulation, two prototypes with different impeller outlet width were manufactured and tested. The variation trend of pump performance with changing impeller outlet width was presented after a series of systematical analysis and contrast.

2 Geometric Model

2.1 Design parameters

The commonly used deep-well centrifugal pump of 150QJ20 type was selected as the research object. Its main parameter at the design condition as follows: rated flow Q_N=20 m^3/h, single-stage head H_s=13 m, stage number N=5, speed n=2 850 r/min, specific speed n_s=113.

This paper defines the single-stage head H_s as follows:

$$H_s = \frac{p_o - p_i}{\rho g N}, \tag{1}$$

where p_o is the outlet total pressure for one section, p_i is the inlet total pressure of one section, and N is the stage numbers in one section.

2.2 Impeller design

The cross-section of 150QJ20 type deep-well centrifugal pump is indicated in Fig. 1.

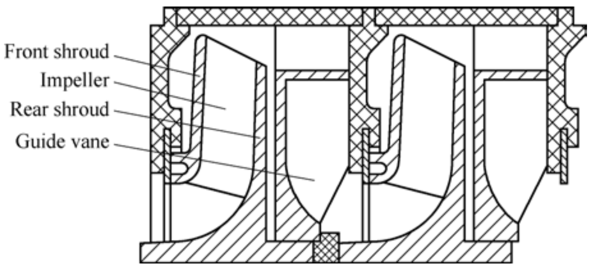
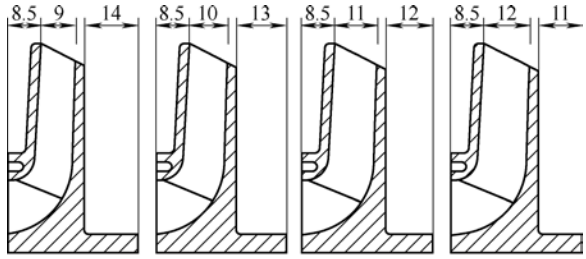


Fig. 1. Cross-section

The impeller's front shroud is extended to the pump casing diameter approximately to maximize the single-stage impeller head[12]. Table 1 shows the main geometric parameters of the impeller. The changing value of the impeller outlet width (b_2) is 9 mm, 10 mm, 11 mm, and 12 mm, as the impeller cross-sections shown in Fig. 2.

Table 1. Main characteristics of impeller

Parameter	Value
Blade number z	7
Blade inlet angle β_1/(°)	26
Blade outlet angle β_2/(°)	22
Blade wrap angle φ/(°)	130
Blade outlet width b_2/mm	9, 10, 11, 12
Front shroud diameter D_{2max}/mm	124
Rear shroud diameter D_{2min}/mm	112



(a) b_2=9 (b) b_2=10 (c) b_2=11 (d) b_2=12

Fig. 2. Impeller cross-section with different outlet width

3 Numerical Simulations

3.1 Calculation model

The calculation model was created based on the real machine, and the whole flow domain consists of 4 components, namely inlet section, impellers, diffusers, and outlet section, as shown in Fig. 3. The outlet section extended to 2 times the impeller diameter length, that the flow could be fully developed at the outlet[13]. After modeled in Pro/E, the assembly model was imported to Gambit for a further processing.

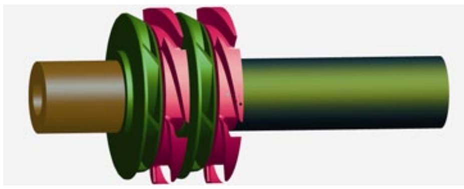


Fig. 3. Sketch of flow domain

3.2 Mesh and grid sensitivity analysis

The whole mesh generation process was carried out in the Gambit 2.2 software, in addition to the inlet section and outlet section were meshed structured girds, two stages whole flow field were meshed unstructured grids. In theory, with the increase of the grid number, the error caused by the grid will be reduced gradually until it disappears. However, considering the configuration of the computer and computing time, the number of grid can not be too large. In this paper, considering the value of y^+ is between 30 and 500[14], so five different grid numbers were selected for the numerical simulation, the numerical results were indicated in Table 2.

Table 2. Grid sensitively analysis

Grids size	1.2	1.4	1.6	1.8	2.0
Grids number	2 423 852	1 634 450	1 125 347	778 993	598 217
Efficiency $\eta/\%$	68.31	68.28	68.24	68.11	67.97
Single stage head H_s/m	14.24	14.23	14.20	14.13	14.06

Table 2 shows the results of grid sensitivity analysis. When the grids size is less than 1.4 or grids number is larger than 1.63 million, the single-stage head and the efficiency experience slight change, indicating that the numerical simulation results are tending towards stable. Considering the computer's calculation capability, the grids size of 1.4 is selected to carry out the following study. Fig. 4 gives a general view of the mesh in the impeller and guide vane.

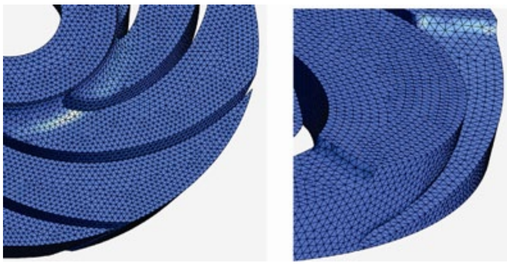


Fig. 4. Sketch of the unstructured mesh

3.3 Boundary conditions

The whole hydraulic passage of the two stages deep-well centrifugal pump was taken as the computational flow domain. The flow domain is divided into 2 types of sub-domains and includes 6 sub-domains, namely, inlet section, first stage impeller, first stage diffuser, second stage impeller, second stage diffuser, and outlet section, among which the inlet section, flow passage of the diffusers, and the outlet section belong to the first type sub-domain. The equations for this type region are solved in a stationary framework. The second type sub-domain is the impeller passage, which is attached to the rotating frame and solved in a rotating framework via the multiple reference frame (MRF), and the rotational speed was set as 2 850 r/min. The interfaces were formed between the different regions.

A uniform axial velocity based on the mass-flow rate is specified at the inlet, and the outlet boundary was assumed to be outflow. At the exit pipes, there is an unavoidable effect on the final flow solution as a result of the boundary conditions. A reasonable length added to the real machine geometry to avoid this effect as much as possible. So at the outlet, which is roughly two impeller-diameters downstream of the vane trailing edge, the gradients of the velocity components are assumed to be zero. The standard wall function was approached to the turbulent flow of near-wall, and all physical surfaces of the pump were set to be no-slip wall.

3.4 Turbulence model

In general terms, the turbulence model describes the distribution of the Reynolds stresses in the flow domain. All turbulence models in use are of an empirical nature. So there is no universally valid turbulence model which will yield optimum results for all applications[15]. Instead it is necessary to select the turbulence model most suitable for the components to be calculated and to carefully validate it by comparing the numerical results with test data. Five turbulence models were adopted to carry out the numerical flow calculations, namely, standard k-ε model, RNG k-ε model, realizable k-ε model, standard k-ω model and SST k-ω model.

Table 3 compares the test and numerical results with different turbulence models. All of the numerical results of five turbulence models are higher than test data, among which the results of standard k-ε model are closest to the test data. Thus the standard k-ε model is chosen for the following numerical calculation.

Table 3. Numerical results with different turbulence models

Turbulence model	Efficiency $\eta/\%$	Single stage head H_s/m
Standard k-ε	68.28	14.23
RNG k-ε	69.96	14.51
Realizable k-ε	68.86	14.39
Standard k-ω	69.83	14.45
SST k-ω	70.14	14.53
Test data	64.67	13.13

3.5 Numerical algorithm and other parameters

The flow through the modeled pump was simulated with the commercial code Fluent, which uses the finite volume method to solve the Reynolds averaged Navier-Stokes equations for 3D incompressible steady flow. The flow model was complemented with a standard k-ε model and logarithmic-law functions for the near wall flow, consistent with the non-slip wall condition. Second order upwind discretizations were used for the convective and the diffusive terms. The pressure-velocity coupling was calculated by means of the SIMPLEC algorithm, and the convergence precision is set to 10^{-5}. During the numerical study, the guidelines proposed in Ref. [16] were used and the numerical uncertainty was related to the change in certain reference values when different mesh refinements were considered. In other words, the values obtained for accuracy can be considered reasonable as to validate the numerical results.

4 Numerical Results and Discussions

4.1 Performance analysis

The pump performances with different impeller outlet width were obtained by numerical simulation under multi conditions. The single stage head, single stage power, and efficiency were shown in Fig. 5. Along with the increase of impeller outlet width, single stage head and single stage power are both increasing gradually, the best efficiency point (BEP) offset to larger flow rate, but the efficiency value at BEP is reduced. Besides, the overlarge b_2 lead to Q-H-curve and Q-η-curve becomes steep, but Q-P-curve becomes flat. It is signify that shaft power is increasing at the same flow rate. In this case, overload phenomenon more likely to be emerged when the pump was running at the relative large flow region.

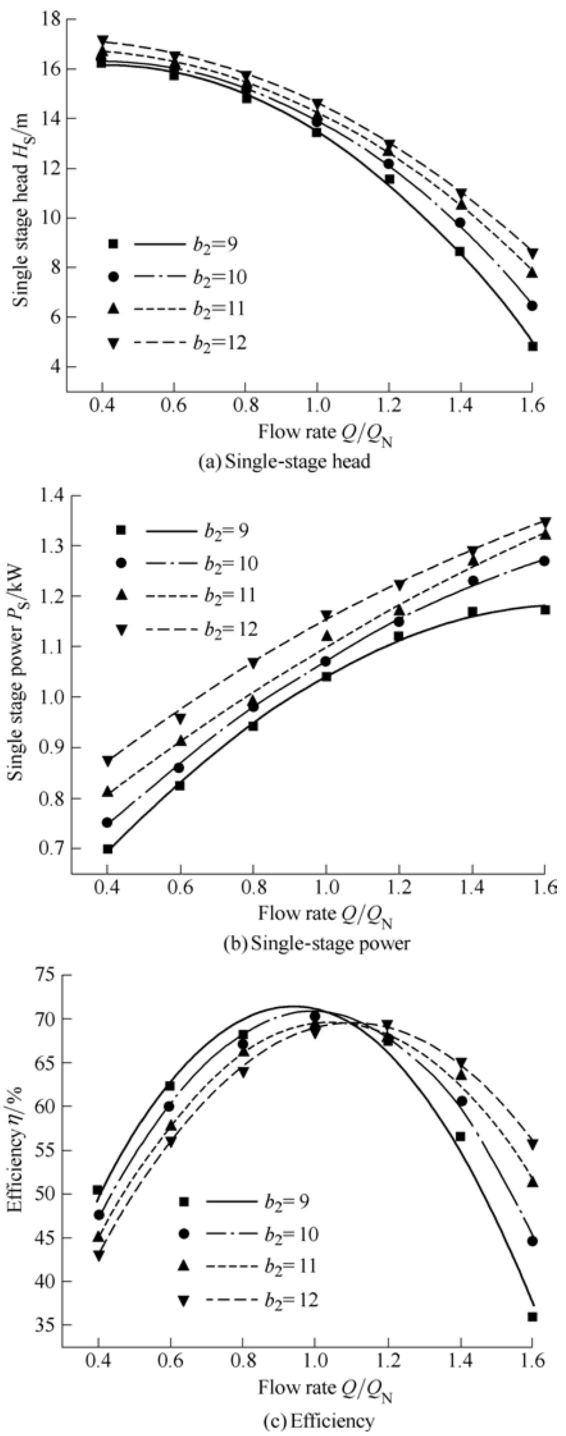


Fig. 5. Pump performance predictions by numerical simulations

The flow rate and pump efficiency at BEP with different impeller outlet width was summarized in Table 4. From 9 mm to12 mm of impeller outlet width, the flow rate of BEP increased about by 18%, and the efficiency of BEP reduced by 3% approximately. Meanwhile, the breadth of high efficiency region has an obvious augment.

Table 4. Flow rate and efficiency at BEP

Blade outlet width b_2/mm	9	10	11	12
Flow rate Q/Q_N	0.93	0.99	1.04	1.10
Efficiency η/%	71.43	70.77	69.61	69.54

The influence of changing impeller outlet width on the pump performance could be indicated form the following analysis. According to flow formula[17]:

$$Q = \pi D_2 b_2 \eta_v \psi_2 v_{m2}, \tag{2}$$

where D_2 is the impeller diameter, η_v is volume efficiency, ψ_2 is the blade expelling coefficient of impeller outlet, and v_{m2} is the meridional component of absolute velocity at impeller outlet. When other conditions remain unchanged, b_2 and v_{m2} is the inverse relationship, which means the change of the impeller outlet width b_2 will cause the v_{m2} change correspondingly. It is important that the change of v_{m2} has a direct influence on the pump performance, as indicated in Fig. 6.

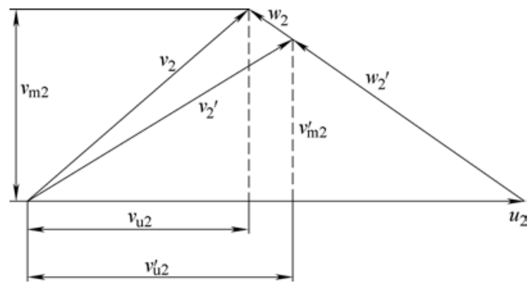


Fig. 6. Velocity triangles at impeller outlet with different outlet width

When b_2 increase to b_2', v_{m2} will be reduced to v_{m2}'. That imply the kinetic energy in the impeller outlet was reduced, and it will lead to the reduction of the hydraulic losses of the guide vane. Besides, it will be caused v_{u2} (i.e., the circumferential component of absolute velocity at impeller outlet) increase to v_{u2}', according to the pump theory head formula:

$$H_t = \frac{1}{g}(u_2 v_{u2} - u_1 v_{u1}), \tag{3}$$

where H_t is the theory head, u_2 is the circumferential velocity of impeller outlet, u_1 is the circumferential velocity of impeller inlet and v_{u1} is the circumferential component of absolute velocity at impeller inlet. Assume that the impeller inlet is irrotational flow, i.e., v_{u1}=0, so

$$H_t = u_2 v_{u2}/g. \tag{4}$$

Therefore, when the other parameters under the conditions and invariable, the increase of impeller outlet width will result in the rise of the theory head accordingly.

4.2 Flow field distribution at design flow condition

Through the numerical calculations under the design flow point, the flow field distributions of different impeller outlet width were acquired. Fig. 7 compares the static pressure distributions at the axial section of impeller. Along with the increase of the impeller outlet width, the increasingly pressure gradient appear at the impeller outlet. In Fig. 7(d) of b_2=12 mm, the uniform of pressure gradient is very obvious and lead to the lager hydraulic losses.

The turbulence kinetic energy contours at the meridian plane of impeller are plotted in Fig. 8. The region of the high turbulence can be observed directly, mainly in the impeller inlet and outlet. In effect, the area and the turbulence kinetic energy of this region are increasing with the increase of impeller outlet width.

The velocity distributions of the impeller meridian plane with four different impeller outlet widths are compared in Fig. 9. It is found that an obvious vortex in the gap of front shroud and pump casing. In Fig. 9(a) of b_2=9 mm, only a few fluids flow into this inner circulation vortex. But as the impeller outlet width increasing, more and more fluids flow into this vortex. Moreover, the streamline in the middle of impeller become bend gradually.

The flow passage in centrifugal impeller is the diffusion type, and controlling the extent of the diffusion is one of the main measures to improve the pump performance. The area ratio of the throat and outlet has an important impact on pump performance. In this study, because of the impeller inlet diameter has no change, the impeller's throat area could be assumed that have not changed. Thus, the increase of impeller outlet width will lead to an increase of the flow area ratio and the decline of pump performance.

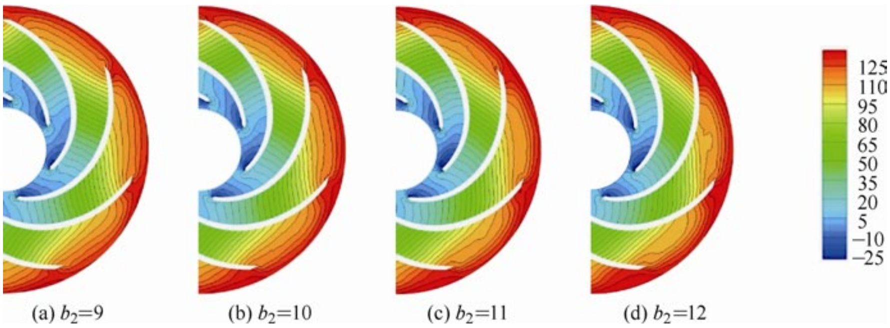


Fig. 7. Static pressure distributions at the axial cross-section of impeller (kPa)

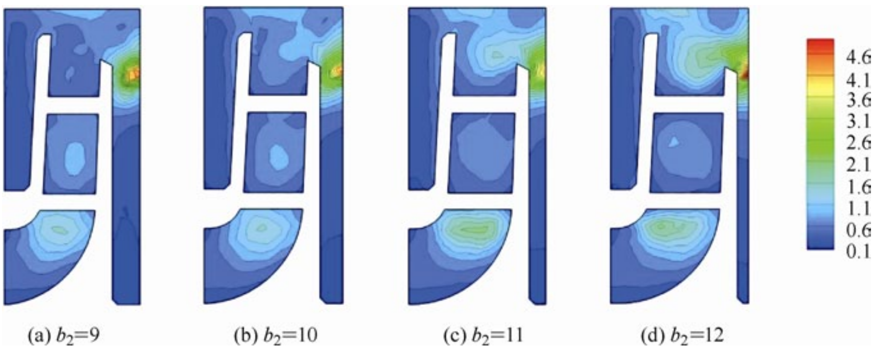


Fig. 8. Turbulent kinetic energy distributions at the meridian plane of impeller (m^2/s^2)

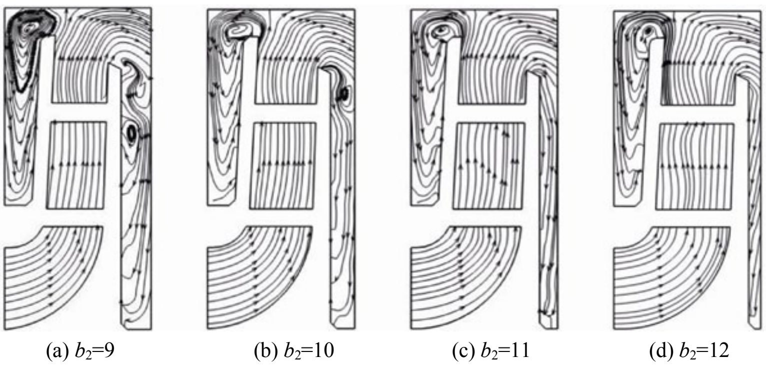


Fig. 9. Velocity distributions at the meridian plane of impeller

5 Test Results and Analysis

In order to further analyze the accuracy of the numerical results, two deep-well centrifugal pumps with different impeller outlet width of b_2=11 mm and b_2=12 mm were manufactured and tested respectively. As shown in Fig. 10, the test-bad has the identification from department of science and technology in Jiangsu province of China, which precedes the requirement of national grade 1 precision (GB/T 3216-2005) and international grade 1 precision (ISO9906-1999)[18].

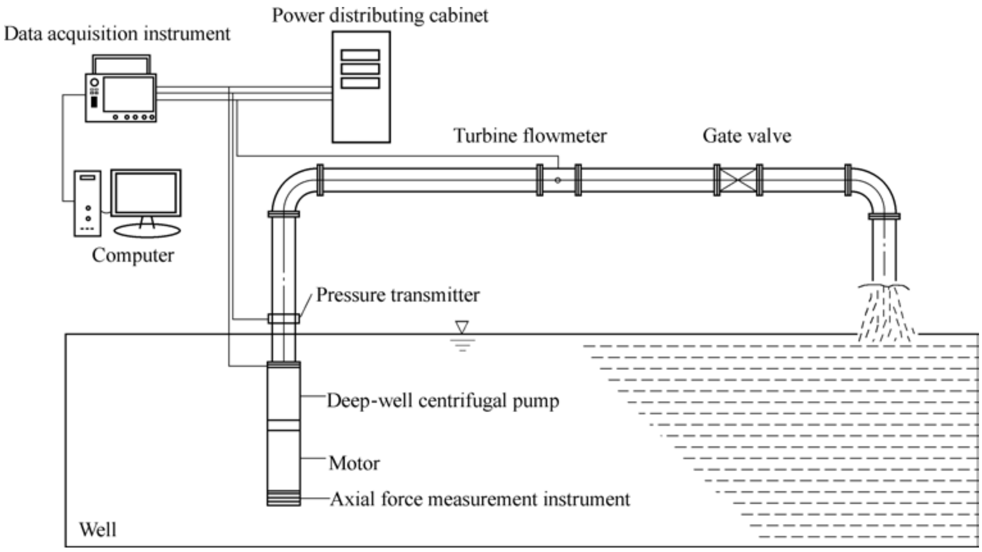


Fig. 10. Deep-well centrifugal pump test rig (not scale)

The pump outlet pressure is measured by a pressure transmitter with 0.1% measurement error. The volumetric flow rate of the pump was measured by a turbine flow meter, and its systematic measurement uncertainty is 0.2%. Torque and speed of rotation were measured by electrical measurement method, and the measurement error of two sensors both is 0.2%. The voltage, amperes and other values of the motor were recorded during the experiments. The overall measurement uncertainty[19] is to be calculated by the square root of the sum of the squares of the systematic and random uncertainties and the calculation result of efficiency uncertainty is 0.5%.

Through the test, the pump efficiency, single-stage head and power of b_2=11 mm and b_2=12 mm were acquired respectively, Fig. 11 show the comparison between test and numerical results. At the rated flow point, both the single-stage head and pump efficiency of numerical results are higher than experimental values, about 8% and 5% severally. The reason may be that the simulation in this paper assumed the face seal in the impeller inlet is completely sealed, neglecting the volume losses of seal leakage. It should be pointed out that, the difference between numerical and test results of two pumps (i.e., b_2=11 mm and b_2=12 mm) are similar. Furthermore, for all the studied flow rates, the numerical results are consistent with the changing trend of the test data, particularly in the operating flow range (80%–120% of the rated flow). This confirmed the feasibility of predicting the deep-well centrifugal pump performance by the numerical method.

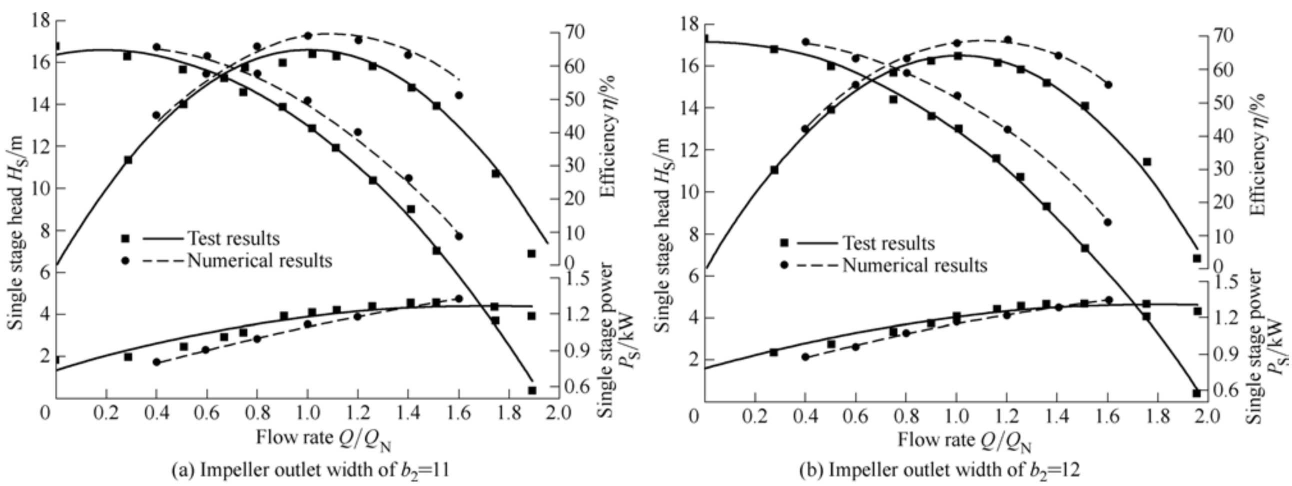


Fig. 11. Comparison of the numerical and test results

6 Conclusions

(1) The numerical results based on the different number of grid and different turbulence model have notable differences. For different geometric model, it is essential to choose the appropriate grid number and turbulence model.

(2) It is one of the most economic and most convenient ways to alter pump performance by changing the impeller outlet width. Along with the increase of impeller outlet width, single stage head and single stage power are both increasing gradually. The best efficiency point offsets to larger flow rate, and the efficiency value at the best efficiency point reduces correspondingly.

(3) An obvious vortex was observed in the gap of front shroud and pump casing. Oversized impeller width leads to the impeller area ratio increasing, and causes the poor pump performance. A point worth emphasizing is that a relative small impeller outlet width conducive to get a better performance and lower power.

References

[1] GOLCU M, PANCAR Y, SEKMEN Y. Energy saving in a deep well pump with splitter blade[J]. *Energy Conversion Management*, 2006, 47(5): 638–651.

[2] MARIO S, HRVOJE K, IGOR S. Improving centrifugal pump efficiency by impeller trimming[J]. *Desalination*, 2009, 249(2): 654–659.

[3] STAN S. When trimming a centrifugal pump impeller can save energy and increase flow rate[J]. *World Pumps*, 1999, 398: 37–40.

[4] TSANG L M. A theoretical account of impeller trimming of the centrifugal pump[J]. *Proceeding of IMechE, Part C: Journal of Mechanical Engineering Science*, 1992, 206(3): 213–214.

[5] TAN Ming-gao, LIU Hou-lin, YUAN Shou-qi et al. Effects of blade outlet width on flow field and characteristic of centrifugal pumps[C]//*Proceedings of the ASME fluids engineering division summer conference*, August 2–6, 2009, Colorado, USA, 2009: 51–60.

[6] REDDY S T C, MURTY R G V, PRASAD M, et al. Experimental studies on the effect of impeller width on centrifugal compressor stage performance with low solidity vaned diffusers[J]. *Proceeding of IMechE, Part A: Journal of Power and Energy*, 2007, 221(4): 519–533.

[7] MAHDI N A, ALI H B, MOHAMMAD D. Investigation of a centrifugal compressor and study of the area ratio and tip clearance effects on performance [J]. *Journal of Thermal Science*, 2008, 17(4): 314–323.

[8] DAI Jiang, WU Yulin, CAO Shuliang. Study on flows through centrifugal pump impeller by turbulent simulation[J]. *Journal of Hydrodynamics, Ser. B*, 1997, 9(1): 11–23.

[9] ZHANG Desheng, SHI Weidong, CHEN Bin, et al. Unsteady flow analysis and experimental investigation of axial-flow pump[J]. *Journal of Hydrodynamics, Ser. B*, 2010, 22(1): 35–44.

[10] JAFARZADEH B, HAJARI A, ALISHAHI M M, et al. The flow simulation of a low-specific-speed high-speed centrifugal pump[J]. *Applied Mathematical Modeling*, 2011, 35(1): 242–249.

[11] 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[J]. *Computer and Fluids*, 2010, 39(5): 859–870.

[12] SHI Weidong, LU Weigang, WANG Hongliang, et al. Research on the theory and design methods of the new type submersible pump for deep well[C]//*Proceedings of the ASME fluids engineering division summer conference*, August 2–6, 2009, Colorado, USA, 2009: 91–97.

[13] WANG F J, LI Yaojun, WANG Yanli. CFD simulation of 3D flow in large-bore axial-flow pump with half-elbow suction sump[J]. *Journal of Hydrodynamics, Ser. B*, 2006, 18(2): 243–247.

[14] WANG Fujun. *Computational fluid dynamics analysis-CFD Principles and application*[M]. Beijing: Tsinghua University Press, 2004. (in Chinese)

[15] GULICH J F. *Centrifugal pumps*[M]. Berlin Heidelberg, Springer, New York, 2007.

[16] CELIK I B, GHIA U, ROACHE P J, et al. Procedure for estimation and reporting of uncertainty due to discretization in CFD Applications[J]. *Journal of Fluids Engineering*, 2008, 130(7): 1–4.

[17] YUAN Shouqi. *Theory and design of low special speed centrifugal pump*[M]. Beijing: China Machine Press, 1997. (in Chinese)

[18] International Organization for Standardization. *ISO 9906 Rotodynamic pumps-hydraulic performance acceptance tests-grades 1 and 2* [S]. British Standards Institution, Geneva, 1999.

[19] FENG H D, XU L, XU R P, et al. Uncertainty analysis using the thermodynamic method of pump efficiency testing[J]. *Proceeding of IMechE, Part C: Journal of Mechanical Engineering Science*, 2004, 218(5): 543–555.

Biographical notes

SHI Weidong, born in 1964, is currently a professor at *National Research Center of Pumps and Pumping System Engineering and Technology*, *Jiangsu University*, *China*. He has published more than 100 papers. His research interests include the theory and design of fluid machinery.
Tel: +86-551-88791266; E-mail: wdshi@usj.edu.cn

ZHOU Ling, born in 1985, is currently a PhD candidate at *National Research Center of Pumps and Pumping System Engineering and Technology*, *Jiangsu University*, *China*. His research interests include the CFD and optimization design of multistage centrifugal pumps.
E-mail: lingzhoo@hotmail.com

LU Weigang, born in 1948, is currently a professor at *National Research Center of Pumps and Pumping System Engineering and Technology*, *Jiangsu University*, *China*. His research interests include the theory and design of pumps.
E-mail: luweigang@ujs.edu.cn

PEI Bing, born in 1984, is currently a master candidate at *National Research Center of Pumps and Pumping System Engineering and Technology*, *Jiangsu University*, *China*.
E-mail: vic_bing@163.com

LANG Tao, born in 1973, is currently an assistant researcher at *National Research Center of Pumps and Pumping System Engineering and Technology*, *Jiangsu University*, *China*.
E-mail: lt800cn@ujs.edu.cn