A Comparative Study of Low-Specific Speed Centrifugal Pump Design Methods

Jianhua Liu *

School of Mechanical and Electrical Engineering, Xinxiang University, Xinxiang,453003, China

***Abstract* — Aim at the problem on low-specific speed centrifugal pump design in different design methods and Unsatisfactory design results, the low-specific speed centrifugal pumps with the same set of parameters was designed by different design methods and experimental Verification was done for every pump. The unsteady three-dimensional numerical simulation was done by CFD in the whole flow field. And the typical flow characteristics inside the impeller and volute were analyzed. The unsteadiness of flow inside the centrifugal pump was confirmed, and flow law of unsteadiness was revealed for low-specific speed centrifugal pump. The research results indicate that the design method to deviate the short blades to the suction surface is very appropriate for low-specific speed pump design and its design effects are ideal.**

Keywords-centrifugal pump; design methods; unsteady flow field; numerical simulation

I. INTRODUCTION

As a kind of universal fluid machinery, the low-specific speed pump is widely applied into aviation, petrochemical, light industry and other fields [1-9], but there is no unified design methodology for low specific centrifugal pump at present in domestic. The most common design methods are the low-specific speed method, the velocity modulus method, the rotor-splitter blades method, etc. Those methods still either base on experience or the results of potential flow analysis[3-10], and the experiments also mainly focus on the external characteristics of the pump[4-12], which lead to the low efficiency, poor cavitation performance and unstable inner flow of the low specific speed centrifugal pumps .this article compared different design methods of centrifugal pumps with the same parameter and adopted the method of the numerical simulation and the experimental verification, respectively, in order to comprehensively understand the whole characteristics of the flow field structure of the centrifugal pumps designed by those methods, capture the characteristics of unsteady flow, provide theoretical basis and design reference for the optimization of future centrifugal pump design.

II. THE DESIGN OF MODEL PUMP

The design parameters of this article are: Q=25m3/h, H=l0m, n=1450r/min. Five schemes are designed according to the low specific speed method, the velocity modulus method and the rotor splitter blades method. To save the production cost of the model pump and shorten the production cycle, these experiments will use the same volute and only change impellers.

A. The design of impellers

In the design process of impeller the main flow parameters were determined firstly , and then the others parameters were determined according to streamline design and structure design. The main flow parameters included inlet diameter of impeller D_j, outlet diameter of impeller D_2, outlet width of blade b_2. The others parameters included blade inlet angle β_1, blade outlet angle β_2, blade quantity Z and scroll of blade φ .The detailed design results of As a kind of universal fluid machinery, the low-specific speed pump is widely applied into aviation, petrochemical, light industry and other fields [1-9], but there is no unified design methodology for low specific centrifugal pump at present in domestic. The most common design methods are the low-specific speed method, the velocity modulus method, the rotor-splitter blades method, etc. Those methods still either base on experience or the results of potential flow analysis[3-10], and the experiments also mainly focus on the external characteristics of the pump[4-12], which lead to the low efficiency, poor cavitation performance and unstable inner flow of the low specific speed centrifugal pumps .this article compared different design methods of centrifugal pumps with the same parameter and adopted the method of the numerical simulation and the experimental verification, respectively, in order to comprehensively understand the whole characteristics of the flow field structure of the centrifugal pumps designed by those methods, capture the characteristics of unsteady flow, provide theoretical basis and design reference for the optimization of future centrifugal pump design.

The impellers can be seen from both figure 1 and figure2

B. The pump body design

The pump body adopted the spiral volute with rectangular cross section in the design of the pump. The diffuser is side outlet. The design result connected by straight lines is as shown in figure 3.

C. The comparative analysis on the design results

To verify the accuracy of the pumps designed conclusion by different methods, the same volute were used to do the external characteristic tests. The experimental results were given in performance curve as shown in the figure 4.

From the performance curve,all the pump performances in different designs can meet the design requirements, the flow head curves of the scheme1, scheme 4 and scheme 5are almost overlapped the flow head curve of scheme 1 drops dramatically and that of scheme 3 is lower. From the efficiency curve, the efficiency of scheme 4 and scheme 5 based on splitter blades is much higher than that of other schemes. The hydraulic efficiency seldom has difference in a small flow condition, but in the big flow condition, the hydraulic efficiency of the scheme 4 and scheme 5 is much higher than others. The result of the analysis is consistent with the analysis of flow field and the analysis of pressure field.

TABLE 1 THE CALCULATION OF IMPELLER DESIGN

Items	Design parameters	Computational formula	Design results
	D_j	$D_j = \sqrt{D_0^2 + d_h^2}$	75[mm]
Scheme1, the low specific speed design method	D_2	$D_2 = 1.05 \times (9.35 \sim 9.6) \times (n_s/100)^{-0.5} \times \sqrt[3]{1.35Q/n}$	188[mm]
	b_2	$b_2 = 0.000375 n_s^{1.15} D_2$	12[mm]
	others parameters	β_1=23.4°;β_2=20;Z=4,φ=160°.	
Scheme2,Speed coefficient design method	D_2	$D_2 = (9.35 \sim 9.6) \times (n_s/100)^{-0.5} \times \sqrt[3]{Q/n}$	180[mm]
	b_2	$b_2 = (0.64 \sim 0.7) \times (n_s/100)^{5/6} \times \sqrt[3]{Q/n}$	9[mm]
	others parameters	β_1=29.5°;β_2=41°;Z=6,φ=80°	
Scheme3,Rotor-splitter blades design method	The blade arrangement adopt Alternative Arrangement structure for three long blade and three short blade, inlet diameter of short blade equal to 0.7D_2 without deflection .		
Scheme4,Rotor-splitter blades design method	Short blade do 10°deflection to pressure surface of the long blade based on the scheme 3.		
Scheme5,Rotor-splitter blades design method	The short blade deflects 10 ° to the suction surface of the long blade based on scheme3.		

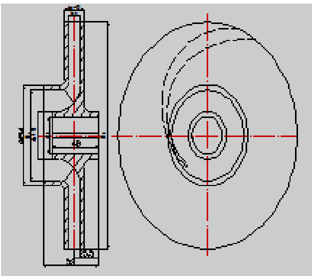

Figure 1. The design structure of scheme1 and 2

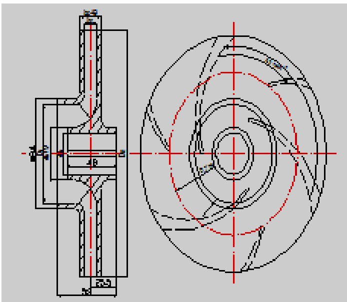

Figure 2. The design structure of scheme 3, 4, 5

DOI 10.5013/ IJSSST.a.17.30.30

ISSN: 1473-804x online, 1473-8031 print

From the performance comparison, the scheme 1 is much easier to realize without overloading performance. The scheme 2 has the worst performance, which is caused by the overcrowding of the blade inlet to the flow. The scheme 3 and scheme 4 have the similar performance except the scheme 4 has better performance in design condition and small flux condition. The scheme 5 is the best one, since it adopts splitter blades and makes the short blades deviate to suction surface.

III. NUMERICAL SIMULATION OF UNSTEADY FLOW FIELDS FOR DIFFERENT DESIGN METHODS

D. The computational domain

The computational domain of the numerical simulation of the centrifugal pump contains inlet and outlet pipe lines, impellers and volutes. To correctly simulate the flow field inside the pump, the physical domain is arranged as following.

The outlet of the impeller is extended without considering the gap among the front and back shroud of the impeller with volutes. Furthermore, in order to maintain the stability and convergence of the flow field inside the pump, the inlet pump is also extended by assuming the outlet of the impeller and the inlet of the volute locate on the same circumference surface. Under the environment of Pro/Engineer, the 3D solid modeling is being processed in the evaluation area of the centrifugal pump based on the five design methods. Then the 3D unstructured mesh is produced by the whole-coupling method in the Gambit mesh generative software, and the local mesh refinement is used around its parts which are respectively next to the impeller area and volute tongue area. Later, the mesh bonding tool, Tmerge software which comes with FLUENT software itself, was used to couple both parts.

E. Control equation

Control equation adopted the Reynolds time averaged N-S equations (RANS), and enclosed by RNG k-ε turbulence model.The control equation can be expressed as follows:

$$\frac{\partial}{\partial t}(\rho u_i)+\frac{\partial}{\partial j}(\rho u_i u_j)=-\frac{\partial p}{\partial x_i}+\frac{\partial}{\partial x_j}(\mu\frac{\partial u_i}{\partial x_j}-\rho\overline{u_i'}\overline{u_j'})+S_i \tag{1}$$

$$\frac{\partial(\rho k)}{\partial t}+\frac{\partial(\rho k u_i)}{\partial x_i}=\frac{\partial}{\partial x_j}(\alpha_k\mu_{eff}\frac{\partial k}{\partial x_j})+G_k-\rho\varepsilon \tag{2}$$

$$\frac{\partial(\rho\varepsilon)}{\partial t}+\frac{\partial(\rho\varepsilon u_i)}{\partial x_i}=\frac{\partial}{\partial x_j}(\alpha_\varepsilon\mu_{eff}\frac{\partial\varepsilon}{\partial x_j})+C_{1\varepsilon}^*\frac{\varepsilon}{k}G_k-C_{2\varepsilon}\rho\frac{\varepsilon^2}{k} \tag{3}$$

In equation, ρ is the fluid density ; u_i is the projection of velocity vector to the direction of i; u_j is the projection of velocity vector to the direction of j; P is the pressure of fluid ; μ is the dynamic viscosity of the fluid; $\rho\overline{u_i'}\overline{u_j'}$ is Reynolds stress; S_i is the projection of generalized fluid to the direction i . k is the turbulent kinetic energy; μ_{eff} is the valid viscosity coefficient; G_k is the formation rate of turbulent kinetic energy k caused by average velocity of gradient. ε is the dissipation rate of turbulent kinetic energy. α_ε , $C_{1\varepsilon}^*$, $C_{2\varepsilon}$ are empirical coefficient.

F. Boundary conditions

During numerical simulation, there mainly exists inlet boundary, export boundary and the solid wall boundary in calculation region. In this paper velocity inlet condition adopted as inlet boundary condition. Export boundary is selected as free outflow boundary condition. Non-slipping wall boundary condition is adopted as the solid wall boundary.

G. The results of numerical simulation and comparison analysis of unsteady whole flow field

1) The distribution hydrostatic pressure of the central cross section under different design methods

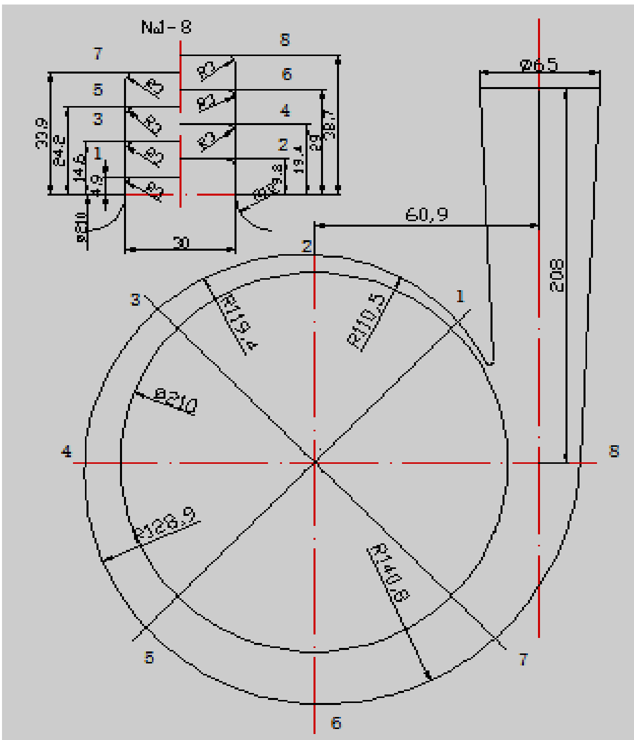

Figure3. Water power of volute designed

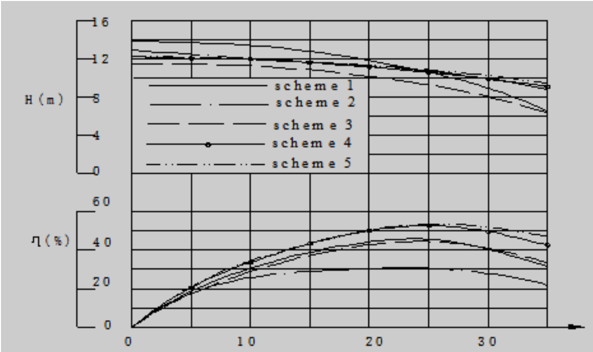

Figure 4. Performance curves of centrifugal pumps by five methods

The figures of the hydrostatic pressure distribution are respectively obtained from unsteady numerical simulation of the pump under five schemes, which set the flow as $25m^3/h$, $10m^3/h$ and $40m^3/h$. It can be seen from those figures that when the flow is low, the pressure fields produced by the rotation of impeller barely change and when the flow increase, the interaction between volute and impellers has a significant influence on the flow field. Compared with other schemes, the differences of internal pressure of the impeller in scheme 5 is much smaller whenever it is the design flow or large flow, therefore, the energy loss caused by the interference of volute and impeller will be lower than other impellers.

2) Pressure fluctuation between internal volute and outlet cross-section based on different design methods.

The hydraulic pressure distribution of different fluxes at different time can only preliminary and qualitatively reflect the changes inside the impellers within different schemes, but the pressure fluctuation between the inlet circumference and the outlet cross section can precisely reflect the in teraction of impellers and volutes. Selecting D=192mm which located in thevolute inlet of the central cross-section as measuring point, Figure 5~ Figure 9 exhibit the fluctuation patterns of hydraulic pressure at different measuring points.

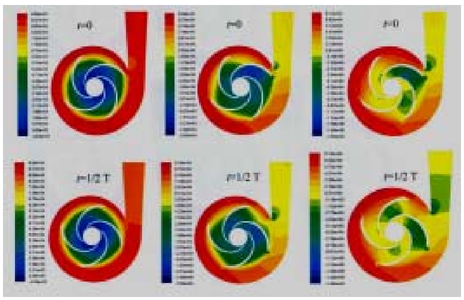

Figure 5. Scheme 1:The distribution of section hydrostatic pressure within different flows [Pa]

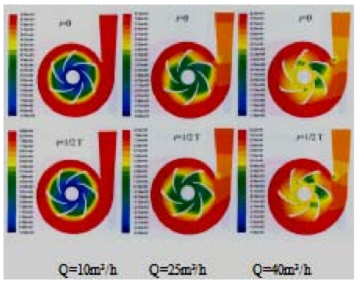

Figure 6. Scheme 2:The distribution of section hydrostatic pressure within different flows[Pa]

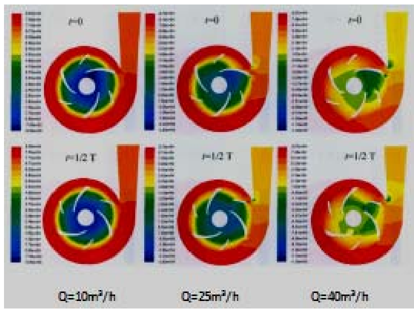

Figure 7. Scheme 3: The distribution of section hydrostatic pressure within different flows [Pa]

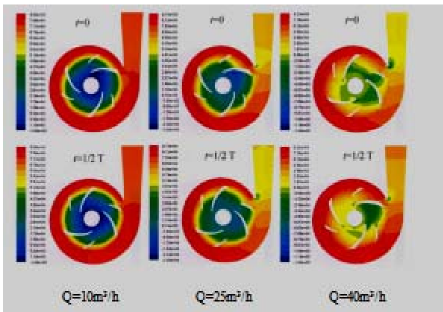

Figure 8. Scheme 4: The distribution of section hydrostatic pressure within different flows [Pa]

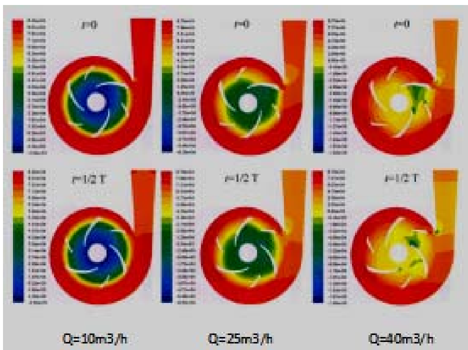

Figure 9. Scheme 5: The distribution of section hydrostatic pressure within different flows [Pa]

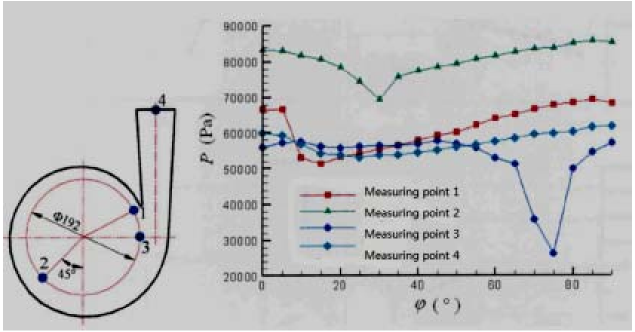

Figure 10. The fluctuation of hydraulic pressure at different measuring points for a cycle of impellers based on scheme 1 （Q=25m3/h).

Figure10 is the changing patterns of the hydraulic pressure from four measuring points obtained from unsteady numerical simulation of the impeller in a cycle based on scheme 1 under the design flow condition.

It can be observed from the figure that with the rotation of the impeller, the pressure fluctuation inside the volute is very significant and the fluctuation range is quite different at different locations. The measuring points 1 and 3 which are next to the volute tongue are greatly influenced by it and the range of pressure fluctuation is much larger than measuring point 2 which is far from volute tongue. The phenomenon indicates that the unsteadiness of the flow is much stronger near the volute tongue. The pressure at the measuring point 1 will drop to the lowest when the blades directly turn to face the volute tongue or when blades pass the volute tongue and the angle is 15°. The pressure of measuring point 3 will drop to the lowest value when the angle is 75°. The pressure of the measuring point 2 located in the middle dropped to its lowest value when the blades pass 25 degree over the volute tongue. As a result, with the rotation of the impellers, the blades continuously pass the volute tongue. The pressure on the different measuring points present fluctuation periodically but with the different location, the range of fluctuation is quite different. Besides the pressure along with the circumference inside the volute, there is also pressure fluctuation on the cross section of volute outlet, although its fluctuation range is much smaller than that on the volute inlet.

From the figure 11, it can be found that when the flow is low, the fluctuation range is small but with the increase of the flow, the fluctuation range becomes larger noticeably. The interaction between impellers and volutes has little effect on the designed condition of low flow, especially on the cross-section of the volute outlet.

3) The comparisons about the distribution of the static hydraulic pressure from five schemes are as followings.

The fluctuation range of the design distribution of the static hydraulic pressure is relatively small in the scheme 3, 4 and 5, but it is significantly larger in the scheme 1.The differences of short blade offset directions will completely make a great difference in flow characteristics. The pressure fluctuation of the scheme 5 in which the short blades deviate to the suction face is far smaller than scheme 3 and 4.

In the figure12, the general tendency of the hydraulic pressure fluctuation on the measuring point of the inlet cross-section based on five schemes was quite similar to measuring point 1 but greatly dropped after being buffered by the diffuser pipe of volute. Similarly, the pressure fluctuation in the scheme 5 is the smallest. From this perspective, the performance of impellers in the scheme 5 is the best.

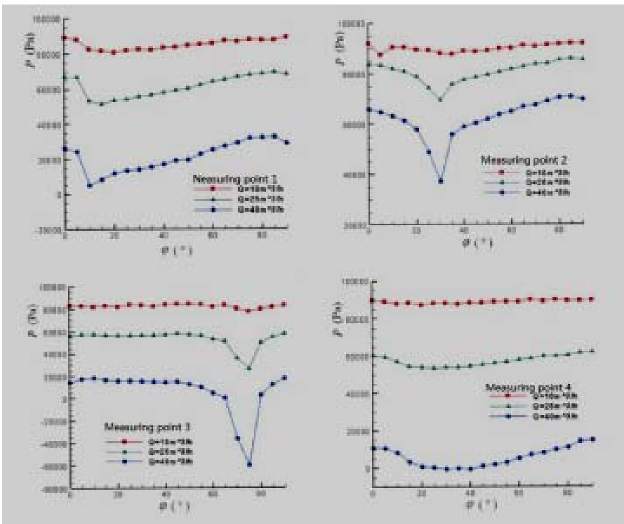

Figure11. The fluctuation of hydraulic pressure at different measuring points for a cycle of impellers based on scheme 1

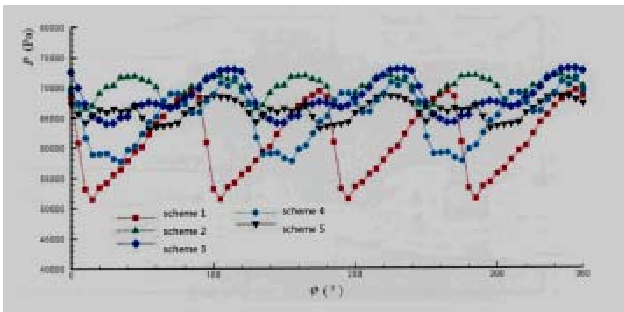

Figure12. The variation of the static hydraulic pressure at the measuring point1 based on different schemes under design condition.

IV. CONCLUSIONS

Focusing on a design parameter at a specific speed of 78, the low-specific speed method, the velocity modulus method and rotor-splitter blades method are respectively used to design several impeller schemes, combining the external characteristic test and numerical simulation of flow field, the schemes are compared and analyzed and the main conclusions are as followings.

(1) The flow shows a strong unsteadiness in the centrifugal pump. Not only at the same time there is a difference of flow structure in each flow channel of impeller, but also the flow changes constantly at different time in the same cannel, revealing a periodic fluctuation. There is periodic pressure fluctuation inside the volute and along with the circumference. The flow unsteadiness near the volute tongue is very prominent.

(2) The amplitude of the pressure fluctuation near the volute tongue is maximum and that on the volute outlet section is minimum; when the flow is small, the flow pressure fluctuation amplitude at each measuring point is relatively small. With the increase of flow, the pressure fluctuation amplitude at each measuring point significantly becomes intense.

(3) For the low specific speed centrifugal pump, the low-specific speed method and rotor-splitter blades method are preferably adopted to design impeller. For the rotor-splitter blades design, the short blade should be used, the desirable inlet diameter should be $0.7D_2$, and the short blades should lean to the suction face of the long blade.

ACKNOWLEDGMENTS

The authors thank the reviewers who gave a through and careful reading to the original manuscript. Their comments are greatly appreciated and have help to improve the quality of this paper.

CONFLICT OF INTERESTS

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

REFERENCES

[1] Zuchao Zhu, The design theory and application of low-specific-speed and high-speed centrifugal pump [M]. Beijing: Machinery Industry Press,2008:65-84.

[2] CHOI Y.KUROKAWAJ.MATSUI J. Performance and internal flow characteristics of a very low specific speed centrifugal pump [J]. J Fluid Eng. 2006. 128(2):341-349.

[3] Songshan Chen, Zhengfu Zhou, Qiang Ge,et al. Orthogonal the experimental study on Splitter blade pump[J]. Journal of Yangzhou University (Nat.Sci.Ed.)2005,8(4):45-48. (in Chinese with English abstract)

[4] Jie Xv, Numerical calculation of the internal flow of complex impeller with low specific speed centrifugal pump [D],Lanzhou:Gansu Agricultural University,2001.

[5] SHIGEMITSU T ,FUKUTOMI J ,KAJI K,et al.Unsteady internal flow conditions of mini-centrifugal pump with splitter blades[J].J Therm Sci,2013,22(1):86-91.

[6] Youshi He ,Shouqi Yuan ,Xiaomei Guo , et, al.Numerical simulation of centrifugal pump impeller with splitter blades with 3D incompressible turbulent flow[J].Journal of the mechanical engineering.2004,40(11):153-157.

[7] Yanning Shen, Shouqi Yuan,Weigang Lu , et, al. Orthogonal test design method based on numerical stimulation for Non-overload centrifugal pump with complex impeller. Transactions of the agricultural machinery. 2010.41(9): 22-26.

[8] Songjin Lian,Songshan Chen ,Zhengfu Zhou , et al. The study of internal flow pattern of t 3D centrifugal pumps with splitting vanes.[J]. Fluid Machinery. 2011.39(3): 18-22.

[9] LIAN Song-jin, CHEN Song-shan, ZHOU Zheng-fu, et al. Study on Three-dimensional Numerical of Turbulent Flow in the Centrifugal Pump with Long and Short Blades, 2011,39(3):18-22. (in Chinese with English abstract)

[10] Chen Songshan, Zhou Zhengfu, He Zhongning, et al. Particle image velocimetry measurement of internal flow characteristics of centrifugal pump impeller with deviated short splitters[J]. Chinese Journal of Mechanical Engineering, 2008, 44(1): 56-61. (in Chinese with English abstract)

[11] Kergourlay G,Younsi M, Bakir F,et al. Influence of splitter blades on the flow field of a centrifugal pump:Test-analysis comparison[J]. International Journal of Rotating Machinery, 2007,: 1-13.

[12] Jianhua Liu, Xiangyang Zhao Miaoxin Xiao . Study on the design method of impeller on low specific speed centrifugal pump [J].The Open Mechanical Engineering Journal， vol9，pp594-600,2015.