The International Journal of Engineering and Science (IJES)
|| Volume || 7 || Issue || 11 Ver.I || Pages || PP 01-09 || 2018 ||
ISSN (e): 2319 – 1813 ISSN (p): 23-19 – 1805

Numerical Investigation Of Internal Flow In A Solid-Liquid Two-Phase Flow Centrifugal Pump

Yu-Liang Zhang[1*], Yan-Juan Zhao[2], Hai-Bing Cai[3], Can-Fei,Wang[3]
1 College of Mechanical Engineering & Key Laboratory of Air-driven Equipment Technology of Zhejiang Province, Quzhou University, Quzhou, 324000, China;
2 College of Information Engineering, Quzhou College of Technology, Quzhou 324000, China
3. Zhejiang Institute of Mechanical & Electrical Engineering Co., Ltd, Hangzhou 310002, China
Corresponding Author: Yu-Liang Zhang

---***ABSTRACT:***---
The solid-liquid two-phase flow centrifugal pumps are widely applied in reality, but the complicated flow in pump makes plentiful of flow rules not be completely revealed so far. Theory research and design method concerning them aren't very mature, so low efficiency, more abrasion and poor reliability always universally occur. To enhance design capability of two-phase flow pump, two-phase flow behavior of 3-D flow field requires to be completely mastered. At present, large numbers of researches mainly focus on a designated geometry relative position between impeller vane and volute tongue, while these flow characters of different relative positions don't be research in detail, especially two-phase flow pump. The mathematical model of 3-D flow field of two-phase flow centrifugal pump is established by using the computational fluid dynamics(CFD)theory in the paper. In calculation region, multi-block mesh technology and local mesh refinement technology are used to catch flow detail. All 3-D flows are simulated by RNG $\kappa-\varepsilon$ *turbulent model and volume of fluid (VOF) method. In solid-wall region, standard wall function is also applied. The coupling between pressure and speed is obtained by SIMPLE algorithm during simulations. The influence of particle diameter and volume fraction on external characteristics is numerically calculated firstly. After selecting typical particle diameter and volume fraction, the two-phase flows of six different relative positions between designated impeller vane and volute tongue are solved later in four-vanes centrifugal pump, and obtained the distribution rules of pressure field, velocity field and solid particles. According to dynamic sediment rule of solid particles, the abrasion rule in solid-wall region is also deduced. Finally, the performance test and cavitation test are also carried out. The numerical calculation results show that the calculation heads will both decrease with the augment of particle diameter and volume fraction, but the influence of particle diameter is more obvious than that of volume fraction. In a cycle of vane sweeping volute tongue, the area of the minimum pressure region presents a evident change process, namely augment earlier before reduction later. When the distance between designated vane and volute tongue is minimum, the abrasion of back shroud would be severest, and the possible cavitation region in pump would be widest. The influence of volute tongue is unconspicuous on the first half fluid than on the latter half fluid in volute, and hydraulic loss in volute mainly comes from the latter half fulid. The abrasion degree on every suction surface is almost invariable, while the difference on every pressure surface is very remarkable. The volute tongue structure is the essential reason for the asymmetry flow field in pump and periodic pulsation of pressure at outlet. Test results show that performance curve hasn't hump, test pump has good operation reliability. High-efficiency region is wide and no overload phenomenon occurs. Cavitation performance is also satisfactory. In a word, above research results have instruction meaning for enhancing efficiency and reducing abrasion.*
KEY WORDS: *two-phase flow; centrifugal pump; numerical simulation; experimental research*

Date of Submission: 05-11-2018 Date of acceptance: 19-11-2018

I. INTRODUCTION

The solid-liquid two phases centrifugal pumps are widely applied in nearly every sector of national economy now. However, low efficiency and poor reliability are always the key technology problems that restrict its development and application. Based on the flow analysis study, The wear characteristics in solid-liquid two phases centrifugal pump is current research hotspot, it can offer some correlative improvement measures.With the development of computational fulid dynamics (CFD) and computer science, the research of inner flow field by numerical simulation technology has now become a important method for optimizing centrifugal impeller and other flow components, which would obviously reduce model manufacture and test times, so it can shorten the exploition period of new productions.

In past years, a large number of researchers mainly concentrated on a fixed geometry position between blade and volute tongue to carry out numerical simulation[1～7]. Only minority researchers studied the flow characteristics of different geometry position in course of rotation, but the calculation medium only includes one kind of fluid[8,9]. In fact, in order to better study the flow characteristics, omnidirectional flow characteristics should be researched. There are two methods available at present.One method is the use of Moving Mesh technology, which would obtain plentiful flow characteristics of any geometry position between blade and volute tongue[10]. Another method is numerical simulations of steady flow at multi-designated geometry locations. In view of the former uses tremendous computer resources, we decided to adopt the latter to calculate the two-phase flow by the software Fluent in this paper. The impeller has four blades, therefor the angle between adjacent two blades is 90°.The flow is solved from original calculation position, every 15°is a calculation position, so there are in all 6 times numerical simulation of two phases flow in a rotation cycle.

II. CALCULATION MODEL AND METHOD

2.1 Main parameters of pump

The model pump is TDT200-600/25,The rated parameters of solid-liquid pump are as follows: Q=557m3/h, H=26m， n=980r/min, NPSH=3.3m, P=75kW, η=80.8%.In this pump, the diameter of impeller D_2=450mm，the diameter of inlet D_j=230mm，the width of impeller inlet b_1=75mm，the width of impeller outlet b_2=64mm， the specific speed n_s=157.5, the numbers of distorted blades Z=4, and these blades are all distorted.

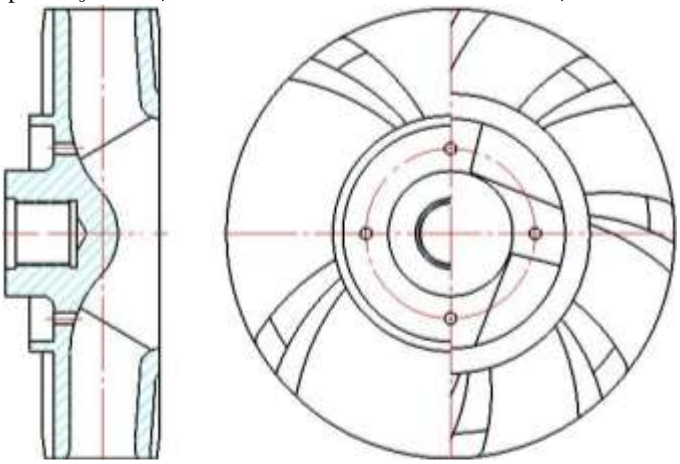


Fig.1 Impeller structure

2.2 Control equations

The 3-D unsteady turbulent flow of incompressible viscosity fluid in pump may be described by Reynold mean momentum equations.

$$\frac{\partial}{\partial t}\left(\rho u_i\right)+\frac{\partial}{\partial x_j}\left(\rho u_i u_j\right)=-\frac{\partial P}{\partial x_i}+\frac{\partial}{\partial x_j}[\mu(\frac{\partial u_i}{\partial x_j}+\frac{\partial u_j}{\partial x_i}-\frac{2}{3}\delta_{ij}\frac{\partial u_l}{\partial x_l})]+\frac{\partial}{\partial x_j}(-\rho\overline{u_i'u_j'}) \tag{1}$$

According to Boussinesq presupposition,the Reynold stress should be as follows:

$$-\rho\overline{u_i'u_j'}=\mu_t(\frac{\partial u_i}{\partial x_j}+\frac{\partial u_j}{\partial x_i})-\frac{2}{3}\delta_{i,j}(\rho k+\mu_t\frac{\partial u_l}{\partial x_l}) \tag{2}$$

In above equations, μ_t is viscosity coefficient of turbulent flow,and is the function of turbulent kinetic energy κ and turbulent dissipation rate ε .RNG $\kappa-\varepsilon$ turbulent model that considering rotation average flow is used to describe control equations, and the equations obout turbulent kinetic energy κ and turbulent dissipation rate ε are as follows.

$$\frac{\partial(\rho\kappa)}{\partial t}+\frac{\partial(\rho u_j\kappa)}{\partial x_j}=\frac{\partial}{\partial x_j}(\alpha_\kappa\mu_{eff}\frac{\partial\kappa}{\partial x_j})+G_\kappa-\rho\varepsilon \tag{3}$$

$$\frac{\partial(\rho\varepsilon)}{\partial t}+\frac{\partial(\rho u_j\varepsilon)}{\partial x_j}=\frac{\partial}{\partial x_j}(\alpha_\varepsilon\mu_{eff}\frac{\partial\varepsilon}{\partial x_j})+\frac{\varepsilon}{\kappa}(C_{1\varepsilon}G_K-C_{2\varepsilon}\rho\varepsilon) \tag{4}$$

In above equations, G_k is turbulent kinetic energy that resulted from gradient alteration. μ_{eff} is equivalent viscidity coefficient, α_κ , α_ε , $\alpha_{1\varepsilon}$ and $\alpha_{2\varepsilon}$ are respectively turbulent model coefficients that determined by experience values.

2.3 Calculation method

According to the flow character in pump, the liquid phase (water) is assumed to be incompressible. The solid phase (sands) is assumed to be the uniform sphere particle, and physical characteristics of each phase in course of flow is also assumed to be unchangeable.Due to its preferable adaptability, the multi-block grid technology is used to respectively mesh impeller and volute, and the two regions are both non-structural tetrahedron grid. the total number of meshes is 940,000, the mesh region is shown in figure 2. Moving Reference Frame (MRF) is used to establish the moving coordinate system that rotates synchronously with the impeller and the static coordinate system that fixes at the volute.The velocity is conversed into absolute velocity form, which may accomplish the data exchange in the junction of the two regions. A mixed model (mixture) is adopted to describe the two-phase flow.The control equations are discrete in space, and every discrete equation embodies all flow parameters in any a calculation cell. In order to insure the calculation convergence, the momentum eqation, the turbulent kinetic energy and the turbulent dissipation rate all adopt the first order upwind difference scheme, while the pressure in equations adopts the standard scheme to solve. Finally, SIMPLE algorithm is used to couple velocity data and pressure data. In this paper, for the sake of reducing the impact of inlet and outlet on the calculation result, the calculation regions at inlet and outlet are extended at certain extent.

Fig.2 Mesh of calculation region

2.4 Boundary Condition

(1)inlet:According to mass conservation and incompressible assumption, the axial velocity u at inlet would be given, and the velocity direction is vertical to the cross section. Turbulent kinetic energy κ and turbulent dissipation rate ε at inlet would be calculated by the following formulas:

$$\begin{cases} \kappa = \dfrac{3}{2}(\overline{u}I)^2 \\ \varepsilon = C_\mu^{3/4}\dfrac{\kappa^{3/2}}{l} \end{cases} \tag{5}$$

In above formulas, the meaning of other parameters may refer to published documents.

(2)outlet: It is assumed that the flow has fully developed at outlet, namely, the flow parameters are no longer change along the flow direction. It satisfies the second type boundary condition,

$$\frac{\partial \varphi}{\partial \vec{n}} = 0$$

namely

$$\varphi_i = \varphi_{i-1} \tag{6}$$

(3)wall:The standard wall function is used at wall surfaces, and it satisfies the no-slip boundary condition.

III. RESULT ANALYSIS

3.1 External characteristics

Under certain specific numerical simulation condition within working range, the sphere particles of five different diameters were calculated respectively under the same volume fraction 5%.The density of solid particles is 2500kg/m3, and the diameters of these particles are respectively 0.1mm，0.3mm，0.5mm，0.7mm and 1mm.

Then under the same particle diameter(0.5mm), five differnet volume fractions are respectively 1%，3%，5%，7% , 10% to be calculated under steady state. The results of numerical calculation are shown in figure 3.

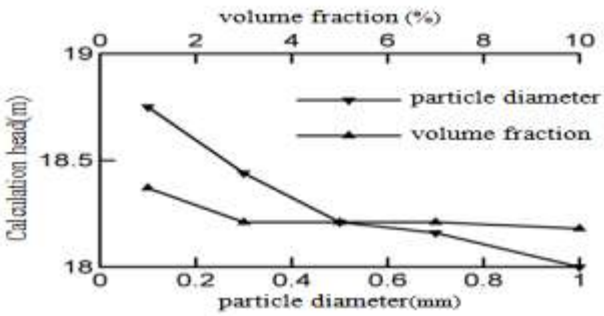


Fig.3 Influence of diameter and volume fraction on head

Figure 3 clearly shows that with the augment of particle diameter and volume fraction, the calculation heads will both present the trend of reduction. But the influence of particle diameter on calculation head is more obvious than volume fraction. Preliminary analysis thinks that with the augment of particle diameter, the required energy that particles maintain suspension and drift maybe become more, so the hydraulic loss would become more in this course, which will result in more obvious reduction of calculation head. At the same time, figure 3 also shows that the influence of volume fraction on calculation head is not quite obvious within certain range. The reason why above rusults appear perhaps is that the liquid phase can produce pressure, while the solid phase can't produce any pressure in flow.

Based on above results and analysises, the later numerical simulation will choose the particle diameter 0.5mm and the volume fraction 5% as calculation conditions of solid phase.

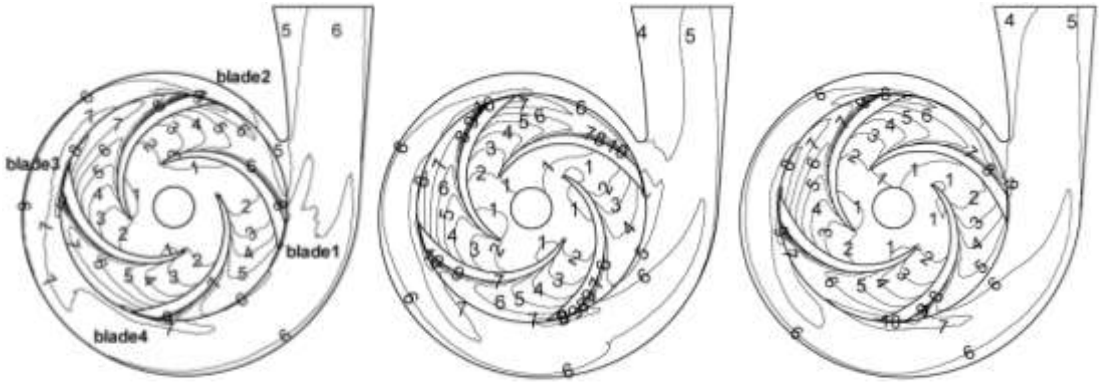
3.2 Pressure distribution

The magnitudes of pressure in pump directly determine the head of pump, and is decisive factor of triumphantly delivering fluid.Besides internal pressure distribution, the minimum pressure region in pump is also research emphasis of cavitation problem. The research of static pressure distribution or dynamic pressure distribution in pump can not both accomplish direct analysis of above problems. On the contrary, the analysis of total pressure distribution can achieve above aims.Figure 4(a-f) show the distribution of total pressure in a cycle at different relative locations between blade and volute tongue. Every blade is marked in figure 4(a), and the marked position in figure 4(a) is regarded as original calculation position in this paper. Every digital representative value in figure 4(a-f) is listed in table 1.

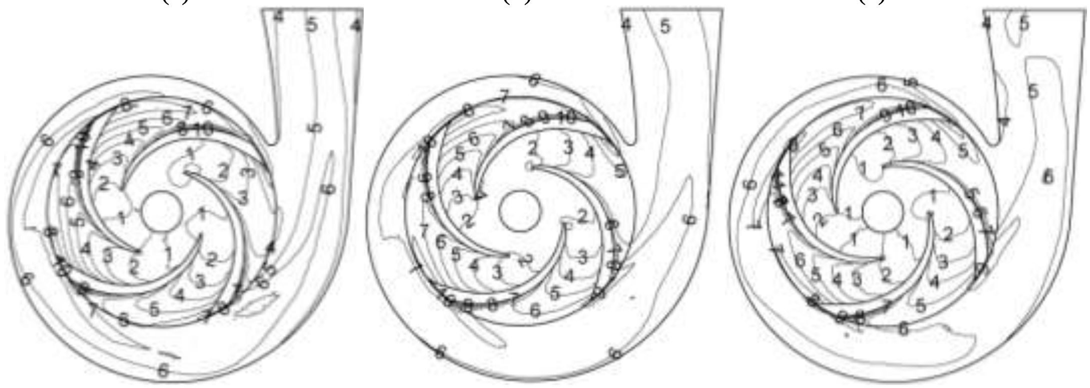
Figure 4 clearly shows that the unsymmetrical geometry structure that volute tongue forms makes pressure distribution also present unsymmetrical distribution structure in pump. Because of the continuous work, with the augment of blade radius, the pressure in each channel also greatly increase, but the pressure gradients present difference in different channels. At the same radius, the pressure on any working surface is obviously higher than the pressure on relevant suction surface. The difference on two surfaces of the same blade just is the ultimate reason for producing lift to drive blade to work.At the same time, the distribution region of the maximum pressure on every working surface mainly concentrates at blade trailing edge. Above result shows that the second half of every blade is mainbody to work for fluid. In a cycle of designated blade sweeping tongue, the minimum pressure region in pump presents a rapid change course, namely augment earlier and reduction later. In this course, when the distance between blade and volute tongue is minimum, the area of the minimum pressure region that locates center region would be maximum. Based on above calculation results and analysises,we may speculate that the area of cavitation in pump is perhaps widest. When the designated blade slowly closes with tongue, the pressure gradient in suction surface channel of designated blade would continuously decrease.On the contrary, the pressure gradient in working surface channel of designated blade would continuously increase. Above all changes are all the influence of tongue on flow in pump, so researchers and designers should strengthen tongue research to improve hydraulic design, which may achieve the dual effect of reducing noise and enhancing hydraulic efficiency.

Table 1 Digital representative value

1	0	7	300000
2	50000	8	350000
3	100000	9	400000
4	150000	10	450000
5	200000	11	500000
6	250000	12	550000



(a) 0° (b) 15° (c) 30°

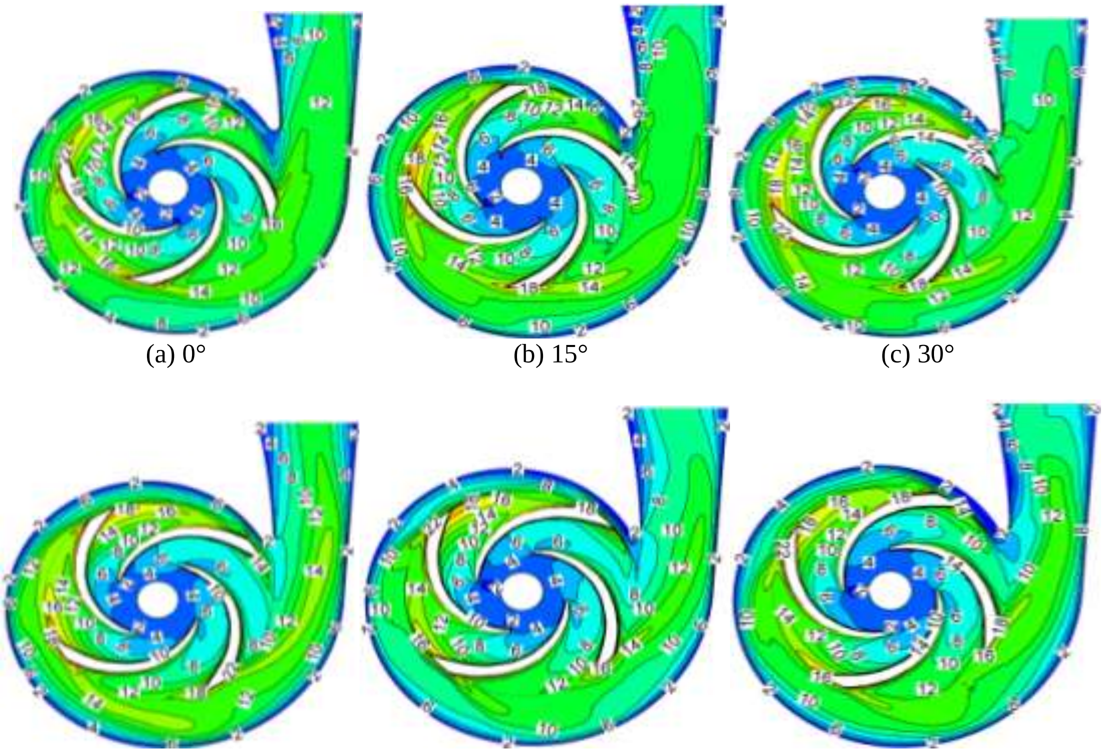


(d) 45° (e) 60° (f) 75°

Fig.4 Total pressure distribution

3.3 Velocity distribution

The magnitudes of velocity in pump directly determine the hydraulic loss of pump, that is to say, the magnitude of efficiency. Because the hydraulic loss is in proportion to velocity square, the distribution and alteration of the maximum velocity in pump should be emphasized. Figure 5(a-f) show that the absolute velocity change in pump as blades rotate to relevant locations, and its unit is m/s. In the entire rotation process, the velocity gradient in the channel that aparts from tongue is more even, and the flow is more calm. Because of its influence, the velocity distribution in neighboring volute channel is also more orderly.We may speculate that the influence of tongue on the latter half fluid in volute channel is more obvious than to the first half fluid, so the hydraulic loss mainly concentrates in the latter half channel.With the decrease of the distance between designated blade and volute tongue, the area of high velocity region in suction surface channel also become wider. When the distance is minimum, the area of high velocity region would be maximum. The internal surface at outlet exists obvious low velocity region.Because of the influence of flow curvature, the area of low velocity region that closes with center line is wider than that velocity region that aparts from center line. As a result of the ununiformity of velocities on cross section at outlet, the pressure difference that velocity difference forms on cross section at outlet results in the secondary flow. At the same time, with the superposition of the secondary flow and the main flow, the spiral forward flow comes into being, which must bring more hydraulic loss. In course of flow, circumferential velociy mainly determines the magnitude of absolute velocity in impeller channels, so the velocity would continuously increase from interior to exterior in every channel.Like pressure distribution, the velocity distribution in pump also presents unsymmetrical structure. As a result of fluid viscidity influence, there is a viscidity bottom layer that has great velocity gradient at inside surface of volute. However the flow velocity is very high in volute, so the thickness of viscidity bottom layer would be very thin. At this time, the roughness degree at inside surface of volute will play the leading role to the turbulent flow core area, so the friction shear stress is mainly brought by turbulent flow Reynolds stress that locates the mainstream region, which would bring more hydraulic loss. Based on above results and analysises, the inside surface of volute should be smooth as possible as to reduce roughness.

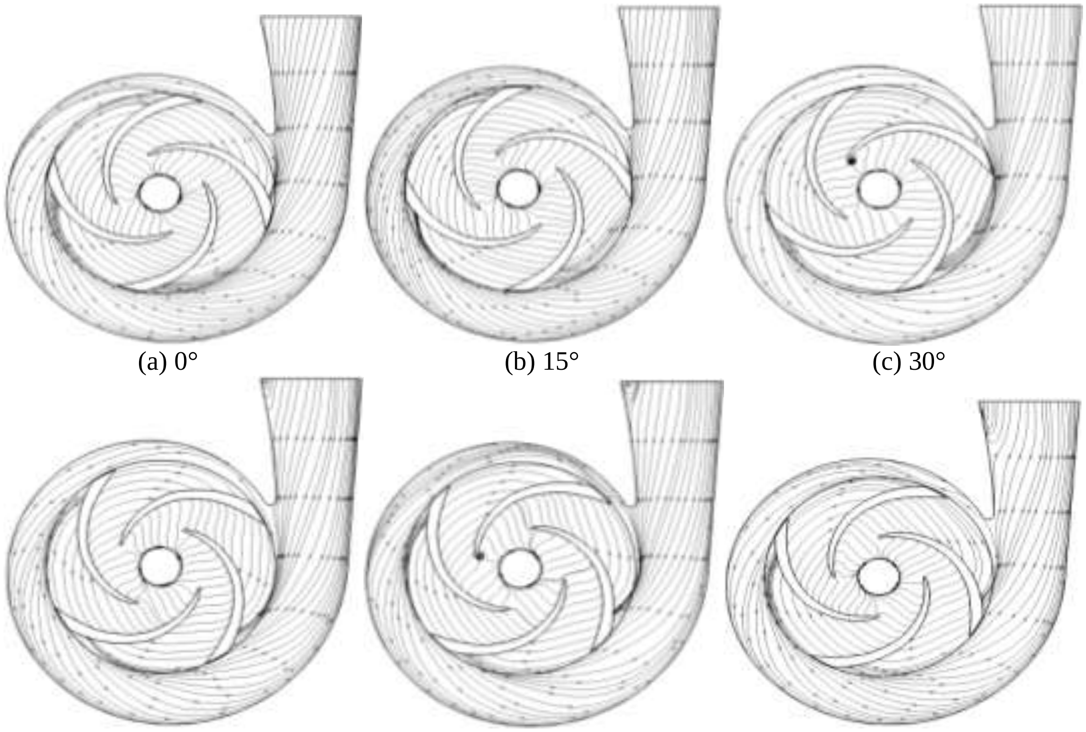


(a) 0° (b) 15° (c) 30°

(d) 45° (e) 60° (f) 75°

Fig.5 Velocity distribution

Streamline distribution are shown in fig.6. Correlative speed mainly determine magnitude of absolute velocity, so streamlines in impeller channel show circle distribution character at any moment. When blade sweeps volute tongue, little area backflow appears at inside position of outlet. According to periodic law, the backflow region soon disappears. When blade rotates to two sides of volute tongue, a low pressure backflow region appears at back surface of inlet of the blade, while no backflow region appears at other part.



(a) 0° (b) 15° (c) 30°

(d) 45° (e) 60° (f) 75°

Fig.6 Streamline distribution

3.4 Particles distribution

The research of particles distribution in solid-liquid pump is very important to research abrasion problem. According to the sediment character of particles, researchers or designers can put forward improving and defending methods to prolong service life. In this paper, when blades are at different rotation positions, the volume fraction of flowing particles on every suction surface is nearly invariable and maintains at about 5%, which is as much as average volume fraction of flowing particles in pump. However, the distribution state of flowing particles on every working surface has remarkable difference as blades are at different rotation positions. The following will take blade 1 as an example to analysis volume fractions of particles at different sites on working surface in a rotation cycle, these sites include the combining site of front shroud and working surface, the combining site of back shroud and working surface, the combining site of half blade height and working surface. Above results may analysis severity degree of abrasion, and provide improvement sake for improving hydraulic design. All calculation results of volume fraction on working surface of blade 1 are shown in figure 7.

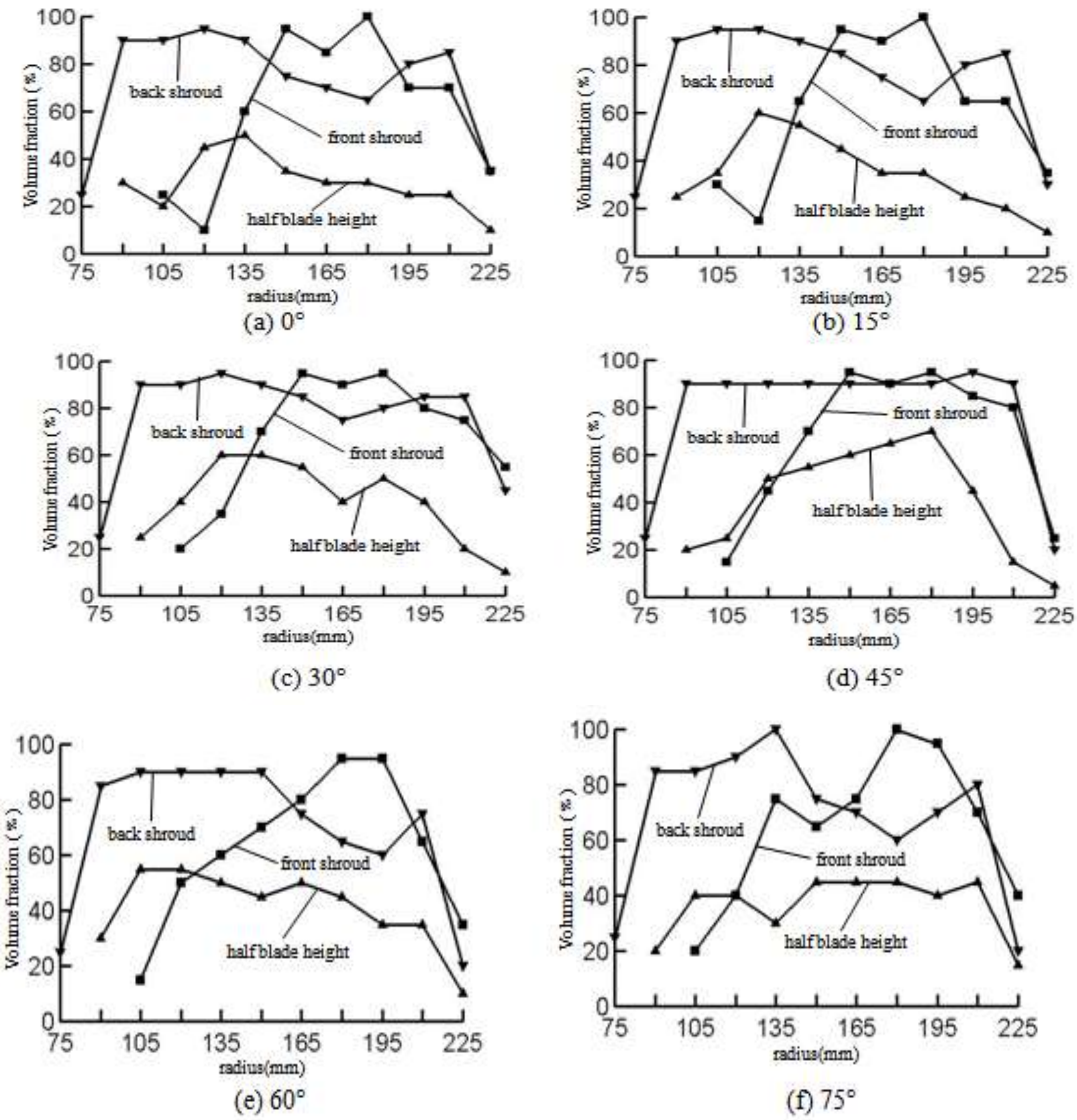


Fig.7 Volume fraction distribution

As a whole, the volume fractions of flowing particles at different sites on working surface have remarkable difference, which manifests that the abrasion degree on working surface is not uniform. This is also the result of unsymmetrical flow that volute tongue forms. In identical rotation position, the volume fractions of flowing particles at front shroud, back shroud and half blade height have also obvious difference. There is a obvious sediment character at mainbody of blade working, namely, the volume fraction of flowing particles at half blade height is less than the volume fractions at back shroud and front shroud, so solid particles at these two combining sites are easier to form flowing abrasion, the abrasion degree is also more severe.In front of blade, the volume fraction of flowing particles at back shroud is considerable relative to the volume fraction at front shroud, so the back shroud is easier to be corraded by solid particles, even worn out. Above this result had been made sure by plentiful tests. Preliminary analysis thinks that because of high density of particles, the inertia is also very much. Except above reason, the curvature radius at back shroud is low, so the two effects bring colliding abrasion at back shroud, which will greatly reduce its intensity at this site. Designers should emphasize the safety measure. At behind blade middle in this example, it is a common tendency that the volume fraction at front shroud is higher than that at back shroud, so the front shroud is earier to be corraded. On the contrary, at blade trailing edge, it is a another tendency that the volume fraction at back shroud is higher than that at front shroud, but almost close at outlet. The reason why the back shroud is easier to be corraded is that the streamline at back shroud is longer, so the accepted energy is also more, and makes the high density particles have more kinetic energy. Above reasons form sucking effect at outlet, which has higher relative velocity and bring sliding abrasion. In the entire rotation process, when the distance between blade and tongue is minimum, the abrasion degree at back shroud is severest, and the corraded area is widest.Therefore the tongue plays a very important role in developing abrasion degree.

3.5 Influence of rotation angle on calculation head

Under certain specific numerical simulation condition within working range, this paper simulated the influence of rotation angle on calculation head in a rotation cycle. The calculation result is shown in figure 8.

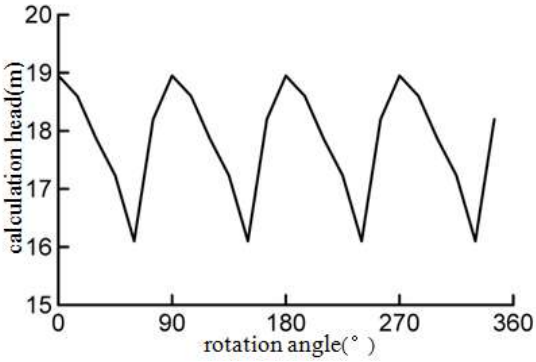


Fig.8 Influence of rotation angle on calculation head

In a rotation cycle, with the continuous decrease of the distance between designated blade and tongue, the total pressue also reduce and reach the minimum at behind tongue.After sweeping tongue, the total pressure start to ascend again. These results manifest that the pressure alteration rule has a repeated course in a entire rotation cycle, namely, period character. Based on above character, we may speculate that the more blades number, the higher pulsant frequency or the shorter pulsant time, which makes pressure more uniform at outlet. All these results further verify the reason why the pressure at outlet presents periodic pulsation, in other words, tongue results in above phenomenon[11,12].

IV. CONCLUSIONS

The two-phase steady flow of six different phases between designated blade and volute tongue in solid-liquid centrifugal pump were solved. The conclusions are as follows. (1)the calculation heads will both decrease with the augment of particle diameter and volume fraction, but the influence of particle diameter is more obvious than that of volume fraction.(2) In a cycle of blade sweeping volute tongue, the area of the minimum pressure region presents a evident change process, namely augment earlier before reduction later. When the distance between designated blade and volute tongue is minimum, the abrasion of back shroud would be severest, and the possible cavitation region in pump would be widest.(3) The influence of volute tongue is unconspicuous on the first half fluid than on the latter half fluid. The hydraulic loss in volute mainly comes from the latter half fulid.(4) The abrasion degree on every suction surface is almost invariable,while the difference on every working surface is very remarkable. In front of blades, the back shroud is easier to be corraded by flowing particles.(5) The volute tongue structure is the essential reason for the asymmetry flow field in pump and periodic pulsation of pressure at outlet.

ACKNOWLEDGEMENTS

The work was supported by Zhejiang Provincial Natural Science Foundation of China (No.LY18E090007) and the National Natural Science Foundation of China (Grant No. 51876103).

REFERENCES

[1]. Jose Caridad, Miguel Asuaje, Frank Kenyery, et al. Characterization of a centrifugal pump impelller under two-phase flow conditions[J].Journal of Petroleum Science and Engineering,2008,6:1-5.

[2]. Rudolf S. Numerical simulation of the two-phase flwo in centrifugal pump imepllers[J].ASME Journal of Fluids Engineering,2001,123(1):1-6.

[3]. GUO Pengcheng, LUO Xingqi, LIU Shengzhu.Numerical simulation of 3D turbulent flow fields through a centrifugal pump including impeller and volute casing[J].Transactions of the CSAE, 2005，21（8）：1-5.

[4]. YUAN Huijing, SHAO Jie, LIU Shuhong,et al. Numerical simulation and LIF-PIV experimental investigation on inner-flow of a mini pump under low flow rate condition[J].Journal of engineering thermo physics, 2008，29(11):1852-1856.

[5]. ZHANG Jing, QI Xueyi, HOU Yihua, et al.Numerical simulation of 3-D turbulent flow fields through double-channel passage impelller[J]. Transactions of the CSAM, 2009，40（1）：64-68.

[6]. Gonzalez j, fERNANDEZ J, BLANCO E, et al. Numerical simulation of the dynamic effects due to impeller-volute interaction in a centrifugal pump[J]. ASME J. Fluids, Eng., 2002, 124:348-355.

[7]. YUAN Shouqi,LI Yi,HE Zhaohui. 3-D calculation of solid-liquid two-phase turbulent flow within a non-clogging centrifugal pump[J]. Chinese Journal of Mechanical Engineering, 2003，39（7）：18-22.

[8]. ZHANG Shujia, LI Xianhua, ZHU Baolin, et al. Applicability of $\kappa-\varepsilon$ eddy viscosity turbulence models on numerical simulation of centrifugal pump[J]. Chinese Journal of Mechanical Engineering, 2009，45(4):238-242.

[9]. YAN Junfeng,CHEN Wei.Numerical analysis of flow features of a high-speed centrifugal pump with a complex impeller with multi-phase position[J]. Journal of Rocket Propulsion, 2007，33(1):28-31.

[10]. XU Zhaohui,WU Yulin,CHEN Naixiang,et al.Simulation of turbulent flow in pump based on sliding mesh and RNG $\kappa-\varepsilon$ model[J]. Journal of engineering thermo physics, 2005，26(1):66-68.

[11]. G.Pavesi, G.Cavazzini, G.Ardizzon.Time-frequency characterization of the unsteady phenomena in a centrifugal pump[J].International Journal of Heat and Fluid Flow,2008,29:1527-1540.

[12]. R.Spence,J.Amaral-Teixeira.Investigation into pressure pulsations in a centrifugal pump using numerical methods supported by industrial tests[J].Computers & Fluids,2008,37:690-704.

Yu-Liang Zhang "Numerical Investigation Of Internal Flow In A Solid-Liquid Two-Phase Flow Centrifugal Pump "The International Journal of Engineering and Science (IJES),), 7.11 (2018): 01-09