The International Journal of Engineering and Science (IJES)
|| Volume || 7 || Issue || 12 Ver.I || Pages || PP 17-24 || 2018 ||
ISSN (e): 2319 – 1813 ISSN (p): 23-19 – 1805

Starting characteristics investigation on a small prototype pump

Yu-Liang Zhang[1*], Zu-Chao Zhu[2], Yan-Juan Zhao[3], Can-Fei,Wang[4], Hai-Bing Cai[4]

1 College of Mechanical Engineering & Key Laboratory of Air-driven Equipment Technology of Zhejiang Province, Quzhou University, Quzhou, 324000, China;
2 The Zhejiang Provincial Key Lab of Fluid Transmission Technology, Zhejiang Sci-Tech University, 310018, Hangzhou, China;
3 College of Information Engineering, Quzhou College of Technology, Quzhou 324000, China
4 Zhejiang Institute of Mechanical & Electrical Engineering Co., Ltd, Hangzhou 310002, China
Corresponding Author; Yu-Liang Zhang

ABSTRACT

To accomplish precise numerical simulation for transient problem in fluid machinery, a precise method was put forward—numerical simulation based on external performance results. Wherein rotational acceleration was taken into account and experimental results was taken as initial condition and boundary condition of numerical simulation, The merits of sliding mesh technique and dynamic mesh technique were fully applied respectively. The object is a low-specific-speed centrifugal pump, whose impeller is open and designed with straight blades.There is a clear head impact phenomenon. Additional head of model pump mainly comes from rotational acceleration and has nothing to do with flowing acceleration. The rise processes of flowrate and head evidently lag behind that of rotational speed. Pressure difference between pressure side and suction side mainly distributes at middle and exterior regions. With the advance of starting time, the distribution area of pressure difference begins to spread towards center and the difference will augment. Rossby dimensionless number will decrease and backflow region begins to appear. The influence of volute tongue on unsymmetrical flow field will be more and more obvious.

KEYWORDS: *centrifugal pump; straight blades; starting; numerical simulation; fluid acceleration*

Date of Submission: 18-12-2018 Date of acceptance: 31-12-2018

I. INTRODUCTION

Centrifugal pumps have been used widely. These pumps usually work under stable operation conditions, namely that rotational speed or load is constant or has tiny variation. By far, many scholars have deeply researched them under stable operation conditions and obtained plentiful achievements[1~7]. With the development of application field, for example, rapid starting, stopping, transient characteristics of centrifugal pump are more necessary to be researched. Unsteady flow may be caused by one or both of following reasons: wall movement and self-inducing fluid. Starting and stopping of fluid machinery belong to the former. Researches showed that transient effect is very obvious during rapid starting period[8~17]. At present, transient researches mainly focuses on water hammer calculation of piping system, while the transient research for only pump is very limited. Tsukamoto[8] made a theoretical and experimental study on the transient characteristics of a centrifugal pump during its rapid acceleration from standstill to final speed, they found that the impulsive pressure and the lag in circulation formation around impeller vanes play predominant roles for the difference between dynamic and quasi-steady characteristics of turbopump. In Lefebvre's study[9], results showed substantial transient effects in overall impeller performance and demonstrated that the quasi-steady assumptions commonly used for the design of impellers that operate under high transient (accelerating or decelerating) conditions are not valid. Thanapandi[11] tested a volute pump with different valve openings to study the dynamic behavior during normal start up and stopping, its results showed that dynamic characteristics severely departure from their steady-state characteristics. Wang and Wu[12~17] adopted different methods to study starting period for centrifugal pump and mixed flow pump. In conclusion, those conclusions they achieved have some differences due to different research objects, methods and test conditions,.

1. Numerical Simulation Method

The interior flow in pump directly determines external performance. When pumps work under unstable operation conditions, it is very necessary to study interior flow field. Pump and piping system together determine a operation point. If the whole system is calculated, it is very difficult to give boundary conditions precisely. In order to eliminate the difficulty, a precise calculation method was put forward---numerical

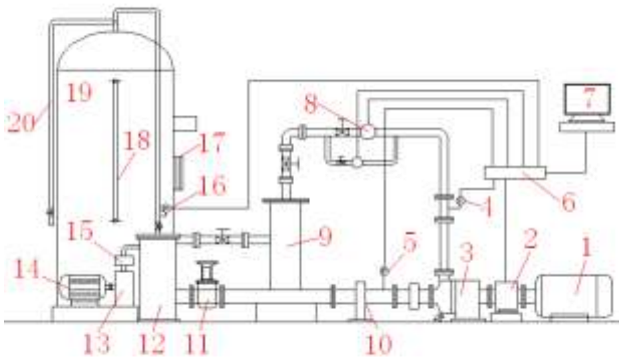
simulation based on external performance results. The method takes into account rotational acceleration and takes experimental results as initial condition and boundary condition of numerical simulation, and then writes to program by user defined function. Wherein rotational speed and flowrate are indispensable.
At present, three methods can be used to accomplish numerical simulation of transient problem. Moving Reference Frame (MRF)[18]—Transient period is divided into finite time interval according to time sequence. In any time interval, numerical simulation is carried out according to quasi-steady assumption. The method doesn't take into account rotational acceleration, and the relative position between blade and volute tongue isn't ascertained accurately. Dynamic mesh method[14]—No any assumptions are used in numerical simulation, so it is the very precise method in theory. The calculation idea is consistent with real rotation status completely. But regenerate mesh magnitude is enormous and mesh distortion is very severe too, which make numerical simulation difficult up to date. With the development of dynamic mesh technique and the enhancement of computer capability, dynamic mesh method will become the preferred simulation method. Sliding mesh method[3,5]—The method is widely used to analyze the unsteady flow under stable operation conditions, especially rotor-stator interaction. Its disadvantage is that rotational speed is invariable, so it isn't competent for transient problem. This paper fully applies the merits of sliding mesh technique and dynamic mesh technique to accomplish precise numerical simulation. Rotational speed could be written to program by user defined function, while sliding mesh technique accomplish slide link between rotor and stator. Rotational law of impeller is consistent with objectivity.
In this paper, the research object is a low-specific-speed centrifugal pump, whose impeller is open and designed with straight blades. Transient effect was investigated by three methods, namely theoretical analysis, hydraulic test and numerical simulation.

II. HYDRAULIC TEST

2.1 Experiment Equipments

Model pump is a small flowrate centrifugal pump, whose impeller is open and designed with straight blades. Design parameters are as below: flowrate is $2.5\,m^3/h$, head is $120m$, rotational speed is $2900\,rpm$. Hydraulic test was carried out on closed test bench with B grade precision. Experiment equipments are shown in Fig.1.

Power source is variable-frequency motor, transient flowrate is measured by LWGY turbine flowmeter, transient rotational speed is measured by JC1A rotation speed sensor and JW-2A microcomputer torque meter. Acquisition and disposal of test datum are accomplished by PCI-6023E data acquisition card and LabVIEW virtual instrument platform[19]. KF1851 electric capacity type differential pressure transmitter is used to measure pressure at inlet and outlet.



1.variable-frequency motor; 2.torque meter; 3.test pump; 4. pressure sensor(outlet); 5.pressure sensor(inlet); 6.data acquisition card; 7.computer; 8.turbine flowmeter; 9.cylindrical stabilizer; 10.locomotive support; 11.gate valve; 12.gas-water separator; 13.vacuum pump; 14.motor; 15.oil-gas separator 16.vacuum manometer; 17.thermometer; 18.level meter; 19.vessel; 20.exhaust pipe

Fig.1 Experimental equipments

2.2 Results and Analysis

Rotational speed accelerates from still to maximal stable value during rapid starting period. When maximal rotational speed is $2930\,rpm$ and valve opening is 1.44 times rated flowrate, the experimental results are shown in Fig.2.

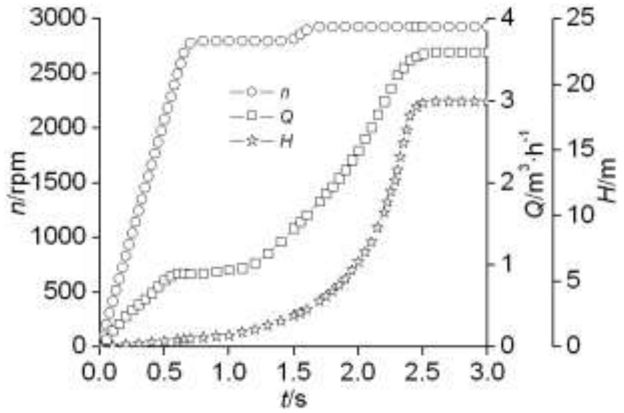


Fig.2 Experimental results

Test results show that it takes about 0.8s to accomplish 95% accelerating process. There is a approximate linear relation between rotational speed and time. Rotation speed reaches maximum 2930 rpm when starting time is 1.7s. The reason for the phenomena is power source—variable-frequency motor, its starting performance directly determines the starting performance. In other words, rotational speed nearly has nothing to do with resistance of piping system. The difference of time histories between flowrate and rotational speed is obvious. Flowrate has a approximate linear relation before 0.7s, and after a transitory smooth process, it increase rapidly again. When time is 2.6s, flowrate reaches maximum. For large valve opening, friction resistance and hydraulic loss of piping system are less, which cause high flowrate. The time that flowrate reaches maximum depends on resistance of piping system. The more resistance is, the less time is. When rotational speed reaches maximum, flowrate doesn't reach maximum. The rise process of flowrate and head evidently lags behind that of rotational speed, which may be directly have relevant to inertia effect of still fluid at the beginning of starting.

III. THEORETICAL ANALYSIS

3.1 Additional Head

Assumptions are as follows: Medium is ideal and incompressible, and no interference among adjacent flow layers each other. According to theorem of moment of momentum, universal expression of transient theory head of centrifugal pump is as below[20].

$$H_d = \frac{u_2 v_{u2} - u_1 v_{u1}}{g} + \frac{\omega}{g Q_d} \times \iiint_\Omega \frac{\partial (v_u r)}{\partial t} \mathrm{d}W_i \qquad (1)$$

where $\mathrm{d}W_i$ is volume element, $\mathrm{d}W_i = r\mathrm{d}\theta \mathrm{d}r \mathrm{d}b$, $\mathrm{d}b$ is thickness of flow layer W_i , ω is transient angular velocity, Q_d is transient flowrate. The first item in Eq. (1) is stable head expressed by flow parameters at inlet and outlet, which is reckoned as the stable head under corresponding rotational speed. The second item is unstable head which takes into account fluid inertia under transient operation conditions, which will bring transient effect. In any transient process, angular velocity (ω) and flowrate (Q_d) are only relate to time(t), while not relate to coordinate. Therefore

$$\rho \iiint_\Omega \frac{\partial (v_u r)}{\partial t} \mathrm{d}W_i = \rho \frac{\mathrm{d}\omega}{\mathrm{d}t} \iiint_\Omega r^2 \mathrm{d}W_i - \rho \frac{\mathrm{d}Q_d}{\mathrm{d}t} \iiint_\Omega \frac{r}{F \cdot tg\beta} \mathrm{d}W_i \quad (2)$$

where

$$\left.\begin{aligned} \rho \frac{\mathrm{d}\omega}{\mathrm{d}t} \iiint_\Omega r^2 \mathrm{d}W_i &= \Omega_J \cdot D^5 \cdot \frac{\mathrm{d}\omega}{\mathrm{d}t} \\ \rho \frac{\mathrm{d}Q_d}{\mathrm{d}t} \iiint_\Omega \frac{r}{F \cdot tg\beta} \mathrm{d}W_i &= \Omega_M \cdot D^2 \cdot \frac{\mathrm{d}Q_d}{\mathrm{d}t} \end{aligned}\right\} \qquad (3)$$

where Ω_J is rotational inertia coefficient of fluid in impeller, Ω_M is flowing inertia coefficient of fluid in impeller, D is nominal diameter of impeller. In centrifugal impeller, $D = D_2$.

$$\left.\begin{aligned} \Omega_J &= \frac{\pi\rho}{32}\left(\bar{D}_2^4\bar{b}_2 - \bar{D}_1^4\bar{b}_1\right) \\ \Omega_M &= \frac{\rho}{8}\left(\frac{\bar{D}_2^2}{\psi_2 tg\beta_2} - \frac{\bar{D}_1^2}{\psi_1 tg\beta_1}\right) \end{aligned}\right\} \qquad (4)$$

where $\bar{D}_1$, $\bar{D}_2$, $\bar{b}_1$, $\bar{b}_2$ are relative dimensions respectively. $\bar{D}_1 = D_1 / D$, $\bar{D}_2 = D_2 / D$, $\bar{b}_1 = b_1 / D$, $\bar{b}_2 = b_2 / D$ .Where D_1 and D_2 are the diameters of inlet and outlet on middle stream surface respectively. b_1 and b_2 are the widths of water-carrying section of inlet and outlet on middle stream surface respectively. ψ_1 and ψ_2 are the crowding coefficients at inlet and outlet on middle stream surface respectively. β_1 and β_2 are the setting angles of blades at inlet and outlet on middle stream surface respectively. So the additional heads of centrifugal pump are as below during transient operations.

$$H_u = H_{u1} - H_{u2} \qquad (5)$$

$$\left.\begin{aligned} H_{u1} &= \frac{\omega}{\rho g Q_d}\cdot\Omega_J\cdot D^5\cdot\frac{\mathrm{d}\omega}{\mathrm{d}t} \\ H_{u2} &= \frac{\omega}{\rho g Q_d}\cdot\Omega_M\cdot D^2\cdot\frac{\mathrm{d}Q_d}{\mathrm{d}t} \end{aligned}\right\} \qquad (6)$$

where H_{u1} is the additional head brought by rotational acceleration, H_{u2} is another additional head brought by flowing acceleration.

3.2 Analysis of Transient Behavior

Main geometry parameters of centrifugal impeller are shown in table1.

Table 1. Geometry parameters of centrifugal impeller

parameter	value	parameter	value
β_1	90°	D_1 / m	0.05
β_2	90°	D_2 / m	0.27
Z	8	b_1 / m	0.017
ψ_1	0.91	b_2 / m	0.005
ψ_2	0.96	D / m	0.27

According to experimental results and geometry parameters, flowing inertia coefficient can be written as.

$$\Omega_M \approx 0 \qquad (7)$$

The conclusion shows that transient behavior of centrifugal pump with straight blades mainly comes from rotational acceleration of impeller and basically has nothing to do with flowing acceleration. Fig.3 and Fig.4 show theory calculation results. In the process of starting, the influence of rotational acceleration on theory head is very evident. The influence of flowing acceleration on theory head is neglectable, or additional head is approximately equal to zero. The centrifugal impeller with straight blades has special 90° setting angles, so the area of water-carrying section increases in the light of approximate linear law. The rise tendency of flowrate is similar to that of the area of water-carrying section, which may lead to tiny head consumption during accelerating period. The rise process of stable theory head is nearly consistent with that of rotational speed. The additional head that rotational acceleration brings mainly generates at the beginning of starting, and reaches maximum when time is about 0.6s. Hereafter, rotational speed continues to rise, but angular acceleration decreases rapidly, which leads to rapid decreasement of additional head and reaches zero when roation speed is invariable. When time is about 1.5s, rotational acceleration appears again and it leads to additional head again. Fig.4 shows that there is a clear head impact phenomenon during rapid starting period, which is consistent with references published[9,15]. Pressure implusion may be the reason for impact at the beginning of starting. The boundary layer is very thin at the beginning of starting, and the flow is regarded as potential flow. Nearly no flow separation appears, this may be also a important reason for presssure impact.

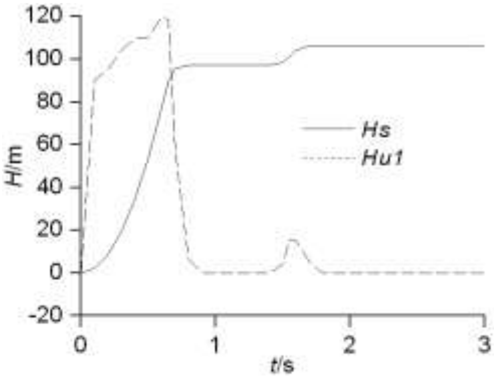


Fig.3 Stable head and additional head

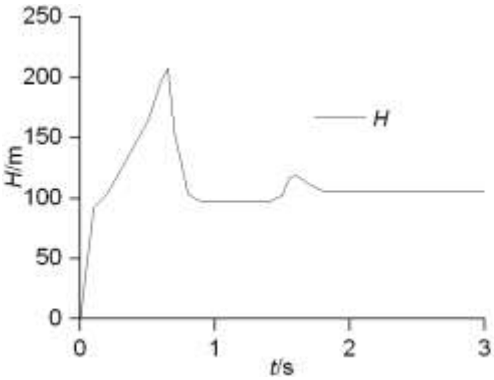


Fig.4 Transient theory head

IV. NUMERICAL SIMULATION

4.1 Calculation control

Unsteady flow was calculated by CFD software —FLUENT that based on finite volume method, Yakhoth and Orzag put forward RNG $k-\varepsilon$ turbulence model in 1986, which has been made sure that it well suited to interior flow of pump and well disposed flow of high strain rate and high curve degree. In calculation region, multi-block mesh technology and local mesh refinement technology are used to catch flow detail, and the total number of grids is 760,000. Open impeller are shown in Fig.5. Time discretization of transient term adopts one order implicit scheme, space discretization of convection term and diffusion term respectively adopts one order upwind scheme and central difference scheme with second order accurate, space discretization of source term adopts standard scheme(linearization). The coupling between pressure and speed is accomplished by SIMPLE algorithm. The convergence criterion is 0.001.

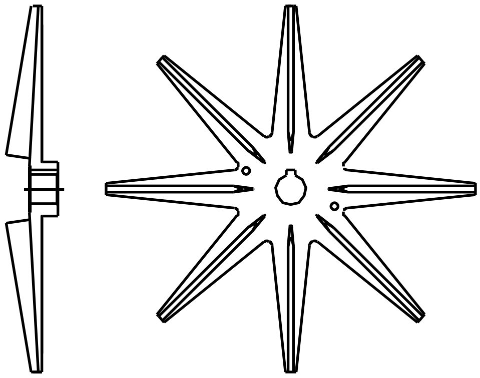


Fig.5 Open impeller

4.2 Boundary condition and initial condition

Boundary conditions and initial conditions should be given simultaneously. To accomplish precise datum fitting, sectional fitting is adopted here.

(1) Rotational speed. The time history of rotational speed is divided into seven segments to fit by polynomial. For example, for 0.6s$< t <$0.8s,

$n = 46670t^3 - 111000t^2 + 87830t - 20320$ (8)

(2) Inlet boundary condition. At inlet, velocity-inlet condition was selected. Time history of flowrate is divided into four segments to fit by polynomial and exponential function. For example, for 0.1s< t <0.6s and 0.6s≤ t < 2.6s,the fitting functions are as follows respectively.

$$Q = -28.12t^4 + 38.31t^3 - 18.35t^2 + 5.068t - 0.1592 \quad (9)$$

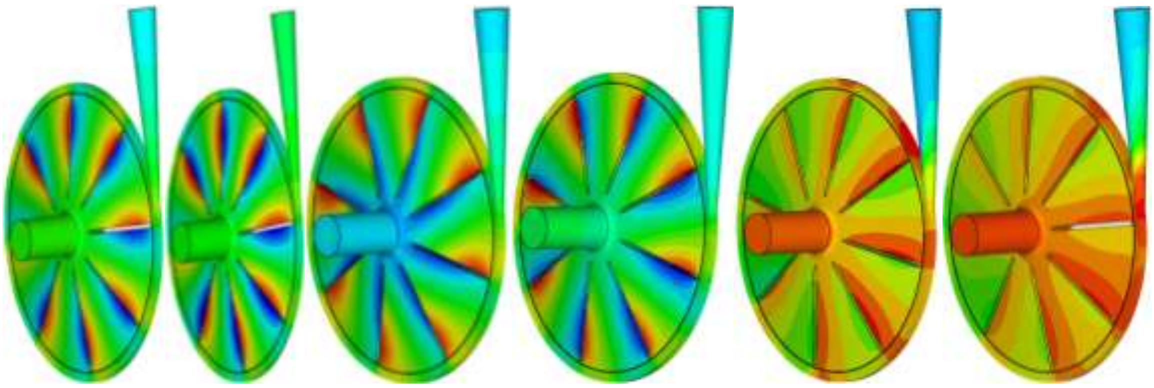
$$Q = 2.704e^{-\left(\frac{t-2.584}{0.7317}\right)^2} + 0.9025e^{-\left(\frac{t-32.67}{401.6}\right)^2} + 0.0954e^{-\left(\frac{t-2.32}{0.1214}\right)^2} + 0.2756e^{-\left(\frac{t-1.61}{0.3408}\right)^2} \quad (10)$$

(3) Outlet condition. At outlet, outflow condition is given.

(4) Wall condition. No slip boundary condition is adopted on wall. Standard wall function is also adopted.

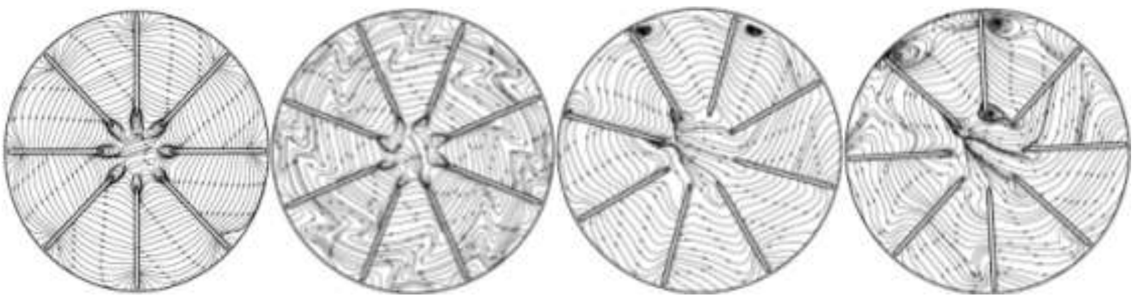
4.3 Results and analysis

Fig.6 and Fig.7 are the distributions of total pressure and velocity field during starting period respectively. For precise rotational speed in simulation, impeller rotates to different positions at different time. The rotational speed is low at the beginning of starting, but some common laws are existent. Pressure on pressure side is obviously higher than that on suction side at the same radius. With the augment of radius, pressure will enhance. Pressure difference between pressure side and suction side mainly distributes at middle and exterior area at the beginning of starting, which may be relate to still fluid. The pressure difference isn't too obvious near central area. With the advance of starting time, the distribution area of pressure difference begins to spread towards center and the difference value begins to augment. Impeller has rotated 1.74 times circle when time is 0.05s. At this time, high pressure regions on pressure side begin to move toward tips of blades, while low pressure region on suction side begins to move toward center. Subsequently, with the augment of rotational speed and flowrate, pressure will enhance as a whole. The influence of volute tongue on unsymmetrical flow field will be more and more obvious.



(a)0.004s (b)0.01s (c)0.05s (d)0.65s (e)2.0s (f)2.6s

Fig.6 Total pressure distribution during starting period



(a)0.004s (b)0.65s (c)2.0s (d)2.6s

Fig.7 velocity field during starting period

Starting period is a transient process, the flow is from laminar flow to turbulent flow. Reynolds number will sharply rise from zero to millions, and hydraulic loss will also sharply rise, therefore flow state is very complex. No backflow appears at the beginning of starting period, which may be relate to still fluid and low rotational speed. Subsequently, the backflow region of low pressure begins to appear at suction side of blades which are located between first section and third section. Rossby dimensionless number can reflect the influence blade curvature and rotation on backflow in impeller., this formula manifests the ratio of inertial force(centrifugal force) to Coriolis force. Wherein curvature brings inertial force and rotation brings Coriolis force. The pump simulated is low-specific-speed, its stable flowrate and relative velocity are both low. While its circular velocity is far higher than its relative velocity, so Rossby dimensionless number is low. With the advance of starting time, it will become lower. At that moment, backflow and secondary flow is easy to appear in calculation region. Except above reason, ring type volute may is one of reasons.

V. CONCLUSIONS

In this paper, the research object is a low-specific-speed centrifugal pump, whose impeller is open and designed with straight blades. Transient effect was researched by theoretical analysis, experimental study and numerical simulation during starting period. Conclusions are as follows.

(1) A precise calculation method was presented—numerical simulation based on external performance results. The method takes into account rotational acceleration and takes experiment results as initial conditions and boundary conditions of numerical simulation. The merits of sliding mesh and dynamic mesh were fully applied.

(2) Experimental study approves that the rise process of flowrate evidently lags behind that of rotational speed.

(3) Theoretical analysis shows that there is a clear head-impact phenomenon. Transient effect of straight blades centrifugal pump mainly comes from rotational acceleration of impeller and has nothing to do with flowing acceleration.

(4) Numerical simulation shows that pressure difference between pressure side and suction side mainly distributes at middle and exterior area. With the advance of time, the distribution area of pressure difference begins to spread towards center and the difference value will augment. Rossby dimensionless number will minish and backflow area begins to appear. The influence of volute tongue on unsymmetrical flow field will be more and more obvious.

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]. ZHU Zuchao, CUI Baoling, LI Yi, et al. Experimental Study on Hydraulic Performance and Wear of Double-channel Pump Delivering Solid-liquid Media[J]. Journal of Mechanical Engineering, 2009, 45(12): 65-69.

[2]. YUAN Shouqi, NI Yongyan, PAN Zhongyong, et al. Unsteady Turbulent Simulation and Pressure Fluctuation Analysis for Centrifugal Pumps[J]. Chinese Journal of Mechanical Engineering, 2009, 22(1):64-69.

[3]. ZHANG Jinfeng, YUAN Shouqi, FU Yuedeng, et al. Numerical Forecast of the Influence of Splitter Blades on the Flow Field and Characteristics of a Centrifugal Pump[J]. Journal of Mechanical Engineering,2009，45（7）：131-137.

[4]. WU Dazhuan, WANG Leqin, BAO Zhiren. Numerical Simulation of Flow in Open-type Mixed Flow Pump and Options of the Discharge Structures[J]. Journal of Mechanical Engineering, 2008，44（8）：90-96.

[5]. ZHAO Binjuan, YUAN Shouqi, LIU Houlin, et al. Numerical Simulation and Particle Image Velocimeter Measurement for Unsteady Turbulent Flow in Double-channel Pump[J]. Journal of Mechanical Engineering, 2009，45（9）：82-88.

[6]. 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.

[7]. Jose Gonzalez, Carlos Santolaria. Unsteady Flow Structure and Global Variables in a Centrifugal Pump[J].ASME Journal of Fluids Engineering, 2006,128:937-946.

[8]. H. Tsukamoto and H. Ohashi. Transient Characteristics of a Centrifugal Pump during Starting Period[J]. ASME Journal of Fluid Engineering, 1982, 104(1): 6-13.

[9]. P. J. Lefebvre and W. P. Barker. Centrifugal Pump Performance during Transient Operation [J]. ASME Journal of Fluid Engineering, 1995, 117(2): 123-128.

[10]. K. Farhadi, A. Bousbia-salah and F. D'Auria. A Model for the Analysis of Pump Start-up Transients in Tehran Research Reactor [J]. Progress in Nuclear Energy, 2007, 49: 499-510.

[11]. P. Thanapandi and R. Prasad. Centrifugal Pump Transient Characteristics and Analysis Using theMethod of Characteristics [J]. International Journal of Mechanical Sciences, 1995, 37(1): 77-89.

[12]. WU Dazhuan, WANG Leqin, HU Zhengyu. Experimental Study on Explicit Performance of Centrifugal Pump During Rapid Starting Period[J]. Journal of Engineering Thermophysics, 2006, 27(1): 68-70.

[13]. HU Zhengyu, WU Dazhuan, WANG Leqin. Transient Hydrodynamic Performance of Centrifugal Pump During Rapid Starting Period: Study of Explicit Characteristics[J].Journal of Zhejiang University (Engineering Science), 2005, 39(5): 605-608, 622.

[14]. WANG Leqin, LI Zhifeng, DAI Weiping, et al.2-D Numerical Simulation on Transient Flow in Centrifugal Pump During Starting Period[J]. Journal of Engineering Thermophysics,2008, 29(8): 1319-1322.

[15]. PING Shiliang, WU Dazhuan, WANG Leqin. Transient Effect Analysis of Centrifugal Pump During Rapid Starting Period[J]. Journal of Zhejiang University(Engineering Science), 2007，41（5）：814-817.

[16]. WU Dazhuan, XU Binjie, LI Zhifeng, et al. Numerical Simulation on Internal Flow of Centrifugal Pump During Transient Operation[J]. Journal of Engineering Thermophysics, 2009，30（5）：781-783.
[17]. CHEN Songying, LI Chunfeng, QU Yanpeng, et al. Transient Hydraulic Performance of a Centrifugal Pump During Rapid Starting Period[J].Journal of Engineering Thermophysics, 2006，27(5):781-783.
[18]. ZHU zuchao, ZHANG Yuliang, LI Yi, et al. Unsteady Numerical Simulation In Two Phase Flow Centrifugal Pump During Starting Process, 9th International Symposium on Experimental and Computational Aerothermodynamics of Internal Flows,2009.
[19]. WU Gang, ZHU Zuchao. Study on Transient Performance of a Pump Used the Technology of Visual Instrument[J].Mechanical & Electrical Engineering Magazine,2003，20(5):177-179.
[20]. CHANG Jinshi. Transients of Hydraulic Machine Installations[M].Beijing: Higher Education Press,2005.

Yu-Liang Zhang" Starting characteristics investigation on a small prototype pump"The International Journal of Engineering and Science (IJES), 7.12 (2018): 17-24