Indexed in
Scopus Compendex and Geobase Elsevier,
Geo-Ref Information Services-USA, List B of Scientific
Journals, Poland, Directory of Research Journals

International Journal
of Earth Sciences
and Engineering

www.cafetinnova.org

ISSN 0974-5904, Volume 09, No. 06

December2016,P.P.2731-2736

Performance Evaluation of Large-Scale Double-Suction Centrifugal Pump based on CFD

XU CUNDONG[1], WEN QINYU[1], DING LIANYING[1], WANG RONGRONG[1] AND CUI LEI[1,2]
[1]School of Water Conservancy, North China University of Water Resources and Electric Power/Collaborative Innovation Center of Water Resources Efficient Utilization and Guarantee Engineering, Zhengzhou, Henan, 450046, China
[2]ShanXi Hydroelectric Investigation & Design Insititute, Taiyuan, Shanxi, 030000, China
Email: zhanghongyang@ncwu.edu.cn

Abstract: This paper studies the declining efficiency of the large-scale double-suction centrifugal pump, which is caused by cavitation erosion and other reasons. The research took 1200s－56 double-suction centrifugal pump as an example, used Pro/Engineer to construct the three dimensional model of its passage flow line, then deployed ANSYS ICEM CFD to divide the mesh, and finally adopted FLUENT for simulation. In using CFD, Reynolds Time-averaged Navier-Stokes formula was used to describe the flow in the water pump, k-ε turbulence model and wall function were used to analyze the three dimensional turbulence, and segregated implicit of pressure coupling equations was used for calculation. The research shows the distribution of flow velocity in the blade, the pressure distribution and the distribution of the full passage flow line. The law of the flow in the large-scale double-suction centrifugal pump is analyzed. By comparing CFD simulation results and test results, it can be perceived that when the design traffic performance is 1.0Q, the error of delivery head and efficiency is the smallest, which is 1.75% and 2.27% respectively. In addition, the efficiency change also tallies with the general working principle of the pump, which suggests that CFD numerical simulation can reflect the internal flow status in the large-scale double-suction centrifugal pump. Therefore, simulation results can be used to modify the pump.

Keywords: *Large-scale double-suction centrifugal pump; internal flow status; cavitation erosion; CFD simulation; performance evaluation*

1. Introduction

In recent years, the research in the three-dimensional turbulent flow field of the water conservancy machinery has been greatly deepened [1], and a number of achievements related to the measurement of flow in it have been made [2-5] with the rapid development in the theory and computer science in numerical simulation of turbulence, especially the development of the CFD software, e.g. FLUENT, STAR-C and CFX-TASC flow, NUMECA. For example, by using CFD to calculate the figure of three-dimensional turbulent flow of the centrifugal pump, Li Cheng Guang and other researchers studied Pressure pulsation characteristics inside of the double-suction centrifugal pump combined k-ε turbulence model and RNG k-ε turbulence model [6]. González and other researchers studied perssure field of double-suction centrifugal pump by numerical simulation and experiment,and compared simulation result and experiment result,comparative test result examine the feasibility of simulating the Pressure pulsation characteristics in the large double-suction centrifugal pump by numerical simulation [7]. Guo Pengcheng and other researchers suggested that utilizing numerical simulation to study the flow field in the impeller would change the traditional way of hydropower design [8]. In terms of the double-suction centrifugal pump, Cheng Yunzhang and other researchers applied FLUENT to studying the flow field in the pump through numerical simulation of turbulence [9], and proved that first, the result produced by CFD could accurately demonstrate the flow features and principles of passage flow line in the double-suction centrifugal pump, and second, the result could guide the hydropower design to modification the double-suction centrifugal pump. With the construction of interbasin water transfer projects, large-scale irrigation and drainage projects and nuclear power plants, the large-scale double-suction centrifugal pump has been widely used. However, there are few researches who study the numerical simulation of it.

In this paper, the 1200s－56 double-suction centrifugal pump used in Jingtaichuan Pumping Irrigation Project in Gansu Province, one of the most common models, was taken as a sample for numerical simulation. The calculation was done with k-ε turbulence model [10-11] established by Pro/Engineer [12] The mesh dividing was conducted by ANSYS ICEM CFD [13-14] and the after processing of the model and numerical simulation of the three dimensional turbulent flow field in the water pump were made through FLUENT [15] The result showed the pressure and velocity distribution of the flow in the centrifugal pump, and the distribution chart of the full passage flow line. This paper also analyses whether CFD can do analog computation of the large-scale double-suction centrifugal pump with big traffic.

2. Governing equation

The flow in the water pump can be regarded as permanent one which cannot be compressed. The strong curvature can be calculated by Reynolds Time-averaged N－S formula as the followings:

$$\frac{\partial E}{\partial \mathrm{x}}+\frac{\partial F}{\partial y}+\frac{\partial G}{\partial \mathrm{z}}=S \tag{1}$$

$$E_x=\left[\rho_u\rho_{uu}-\mu_e\frac{\partial u}{\partial x}\rho_{uv}-\mu_e\frac{\partial v}{\partial x}\rho_{uw}-\mu_e\frac{\partial w}{\partial x}\right]^T$$

$$F_y=\left[\rho_v\rho_{vu}-\mu_e\frac{\partial u}{\partial y}\rho_{vv}-\mu_e\frac{\partial v}{\partial y}\rho_{vw}-\mu_e\frac{\partial w}{\partial y}\right]^T \tag{2}$$

$$G_z=\left[\rho_w\rho_{wu}-\mu_e\frac{\partial u}{\partial z}\rho_{wv}-\mu_e\frac{\partial v}{\partial z}\rho_{ww}-\mu_e\frac{\partial w}{\partial z}\right]^T$$

$$S=\begin{bmatrix}\frac{\partial}{\partial x}\left(\mu_e\frac{\partial u}{\partial x}\right)+\frac{\partial}{\partial y}\left(\mu_e\frac{\partial v}{\partial x}\right)+\frac{\partial}{\partial z}\left(\mu_e\frac{\partial w}{\partial x}\right)-\frac{\partial p}{\partial x}\\ +w_x{}^2-2\rho w_v\\ \frac{\partial}{\partial x}\left(\mu_e\frac{\partial u}{\partial y}\right)+\frac{\partial}{\partial y}\left(\mu_e\frac{\partial v}{\partial y}\right)+\frac{\partial}{\partial z}\left(\mu_e\frac{\partial w}{\partial y}\right)-\frac{\partial p}{\partial y}\\ +w_y{}^2-2\rho w_u\\ \frac{\partial}{\partial x}\left(\mu_e\frac{\partial u}{\partial z}\right)+\frac{\partial}{\partial y}\left(\mu_e\frac{\partial v}{\partial z}\right)+\frac{\partial}{\partial z}\left(\mu_e\frac{\partial w}{\partial z}\right)-\frac{\partial p}{\partial z}\end{bmatrix} \tag{3}$$

In these formulas, ρ represents fluid density; P is fluid pressure including the reduced pressure of turbulence kinetic energy and centrifugal force (unit: Pa); u,v,w are the components of relative velocity in three coordinate axises (unit: m/s); $2\rho w_v$ is the component of coriolis force at x direction, (unit: N); $2\rho w_w$ is the component of coriolis force at y direction (unit: N); μ_e is coefficient of virtual viscosity (unit: Pa/s) which is the sum of viscosity of molecule and Boussinesq eddy viscosityμ_t.

In order to set μ_t, the standard k-ε turbulence model is adopted to devise the closed equations. The representation of the pulsating energy of turbulence k and its dissipation rating ε is defined respectively as:
k-ε equations:

Equation k:

$$\rho u_j\frac{\partial k}{\partial x_j}=\frac{\partial}{\partial x_j}\left[\left(\mu+\frac{\mu_t}{\delta_k}\right)\frac{\partial k}{\partial x_j}\right]+\mu_t\frac{\partial u_i}{\partial x_j}\left(\frac{\partial u_i}{\partial x_j}+\frac{\partial u_j}{\partial x_i}\right)-C_D\rho\frac{k^{\frac{3}{2}}}{l} \tag{4}$$

Equation ε:

$$\rho u_j\frac{\partial \varepsilon}{\partial x_j}=\frac{\partial}{\partial x_j}\left[\left(\mu+\frac{\mu_t}{\delta_\varepsilon}\right)\frac{\partial \varepsilon}{\partial x_j}\right]+\frac{C_1\varepsilon}{k}\mu_t\frac{\partial u_i}{\partial x_j}\left(\frac{\partial u_i}{\partial x_j}+\frac{\partial u_j}{\partial x_i}\right)-\frac{C_2\rho\varepsilon^2}{k} \tag{5}$$

$$\mu_t=C_\mu\rho k^2/\varepsilon \tag{6}$$

$$k=\frac{1}{2}\overline{u_i^{'}u_i^{'}} \tag{7}$$

$$\varepsilon=\gamma\left(\frac{\partial\overline{u_i^{'}}}{x_k}\right)^2=C_D\frac{k^{\frac{3}{2}}}{l} \tag{8}$$

The value of C_1, C_2, c_μ, δ_k, δ_ε in the above equations is similar in various references: C_1=1.44, C_2=1.92, c_μ=0.09, δ_k =1.0 and δ_ε=1.3.

3 Numerical Simulations

3.1 The geometric size and parameter of the original centrifugal pump

Pump 1200s－56 is a double-suction centrifugal pump of big traffic and high efficiency, which is mainly used in carrying water and other liquids having similar physical and chemical characters for water supply in urban areas and irrigation in rural areas. Parameters are listed below:

Inlet diameter of the impeller D_1=730 mm, outlet diameter D_1=730 mm, design traffic Q=10800 m^3/h, delivery head H=56 m, rotation rate n=600 r/min, outlet width of the impeller b=224 mm, the number of blade Z=6, shaft power P=1820 Kw, and efficiency 90.5%.

3.2 Establishing the model

Pro/E is CAD software developed by PTC. Its mechanical design module is an efficient three dimensional mechanical design tool which can build any complex model rapidly, and its data can be processed in mesh dividing software ANSYS ICEM CFD [16]. Thus, Pro/E is adopted to construct the three dimensional model of the runner, suction and volute. Fig. 1 is the photo of the 1200s-56double-suction centrifugal pump, and Fig. 2 and Fig. 3 is the model of the runner and the blade.

Fig.1 *Double-suction centrifugal pump for 1200s －56*

Fig.2 *Three-dimensional model*

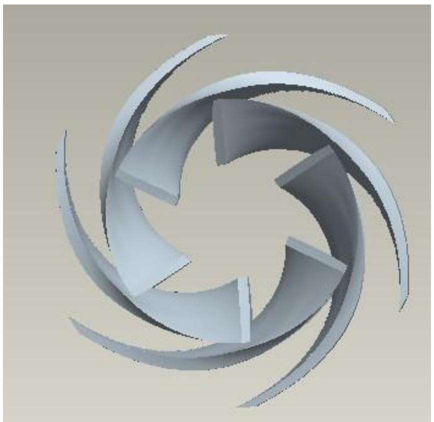


Fig.3 *Blade model of the runner*

3.3 Mesh dividing

CFD can separate the whole area to be calculated to make it a mesh and then discrete the governing equation to it. Specifically, it can transform the partial differential equation into the algebraic equation on every node, and the variable (e.g. speed, pressure) at every node can be calculated through the equation. Thus, the quality of the mesh will have a great influence on the accuracy and the constriction of the calculation and the time spent on it [17].

ANSYS ICEM CFD is used to divide the mesh. The setting can be set to ignore the detail automatically, such as model's geometric defects and insignificant features. The software has some advantages. For instance, it can provide a complete series of tools to restore the model, automatically check the quality of the mesh, do the smoothing process, and redefine the broken unit and revise the mesh visually.

Since the impeller of a large-scale double-suction pump was large and complicated, the mesh dividing was done in the tetrahedron. For the mesh of a complicated geometrical form, especially those parts with high curvature, e.g. the blade, the curved surface of the impeller and runner, dividing the whole area into smaller meshes can reduce the risk of the discrete error. Fig. 4 is the mesh of the runner and Fig. 5 is the mesh of the integral impeller.

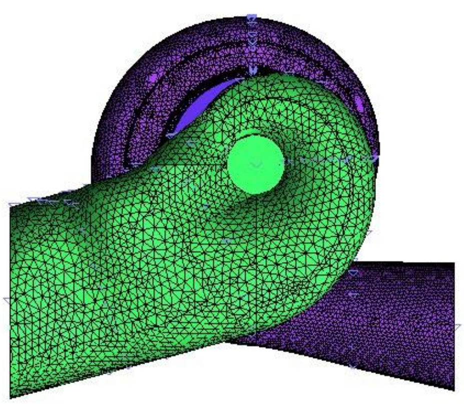


Fig.4 *Mesh dividing of the runner*

Fig.5 *Mesh dividing of integral impeller*

4. Calculation and the boundary condition

The standard k-ε model was applied to the turbulence model, and the standard wall function provided by Fluent was applied to the near-wall surface. The standard wall function was introduced and the influence of the boundary surface was taken into consideration. The area to be calculated contained rotary and static components, and thus it was divided into two parts: the rotary part and the static part for coupling calculation. Moving Reference Frame was deployed. For the flow in the pump could not be condensed or only be slightly condensed, the simulation was conducted by discrete solver with the standard atoms of 101325 P. Gravity was ignored for water was the medium. In the menu Define／Material, the material was defined as water-liquid with the density of 998 kg/m^3.

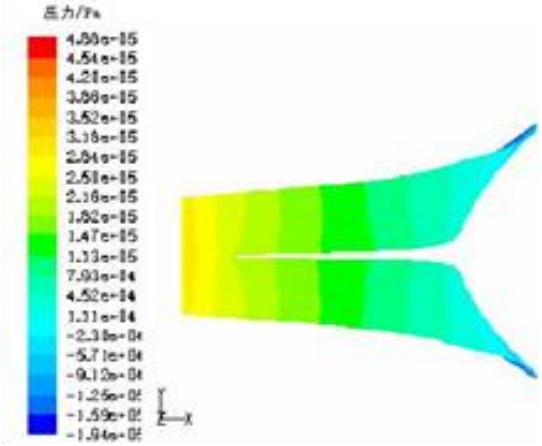
In menu Define／Boundary Conditions, the boundary condition was defined when transporting water as the medium. The discrete approach was a finite volume method. The standard k-ε turbulence model was used for segregated implicit. In calculation, r=const, that is, fluid density was a constant, the inlet and the outlet condition was velocity inlet and free discharge respectively. The condition was non-sliding surface, and surfaces rotate with the runner (such as the crown above, the torus below) were set as being relative static.

In menu Solver/Control/Solution, the interpolation of the pressure was set as the standard format. Energy, momentum, turbulent momentum and dissipation rate were set as relatively stable second order upwind scheme. When in this process, water was supposed to be transported; the coupling of pressure and speed should adopt coordinated SIMPLE, with the iteration accuracy as 10^{-4}. The residual served as the basis for constriction.

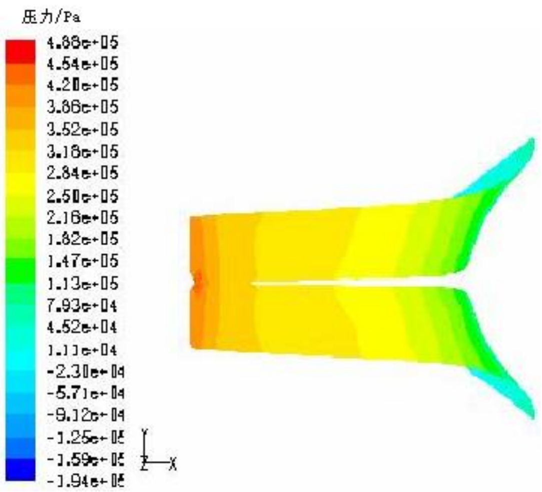
5. Simulation results and analysis

5.1 Simulation results

When water is transported and the design traffic Q=10800 m^3/h, the traffic performance in six conditions, namely 0.2Q, 0.4Q, 0.6Q, 0.8Q, 1.0Q, 1.2Q is analyzed [18] in order to get the delivery head and efficiency under these conditions. The best performance is when the simulation results are as the followings: Fig. 6 and Fig. 7 are the nephograms of the pressure on the working surface and its back, and the distribution graph of the relative velocity. Fig. 8 is the distribution chart of the full passage flow line.

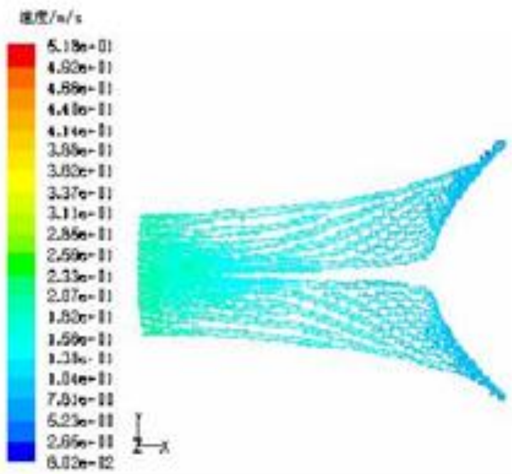


(a) the back of the blade

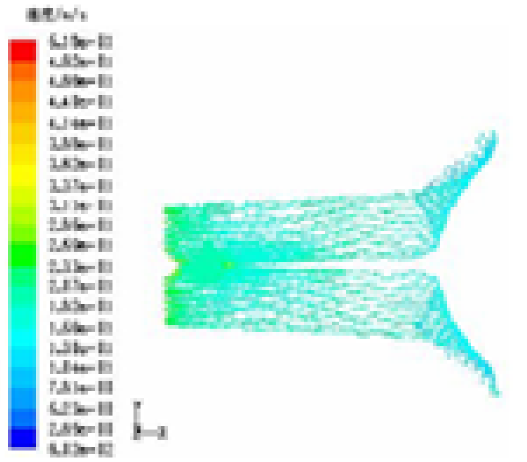


(b) the working surface of the blade

Fig.6 *Cloud chart of blade pressure*



(a) the back of the blade



(b) the working surface of the blade

Fig.7 *Distribution chart of relative velocity at the blade*

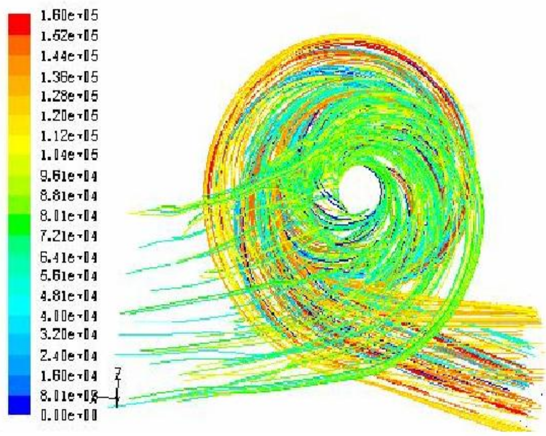


Fig.8 *Distribution chart of the full passage flow line*

From Fig. 6, it can be perceived that the pressure on the blade rises from the inlet to the outlet, and changes dramatically, and most of the kinetic energy is transformed into pressure energy. At the corresponding radius, the pressure on the working surface of the blade is bigger than that of the back, and the negative value of a small part near the back inlet suggests that there is cavitations erosion. From Fig. 7 and Fig. 8, it can be perceived that the flow at the back and the working surface of the blade do not flow inversely or laterally. The flow is smooth-going and the change of its relative velocity with the relative streamline is well-distributed. In addition, the full passage flow line is flow, and the speed distribution is good.

5.2 Comparing simulation and test results

Compare the simulation and test results to examine the feasibility of simulating the flow in the large double-suction centrifugal pump by CFD. Table 1 shows the total pressure values of inlet and outlet simulated by Fluent.

Based on the simulation, water head H and efficiency η_h are worked out by the following formula:

Waterhead:

$$H = \frac{p_{出} - p_{进}}{r g} + Z_{出} - Z_{进} \tag{9}$$

Efficiency:

$$h_{h} = \frac{r gQH}{Mw} \tag{10}$$

P is fluid density (unit: kg/m^3); g is the acceleration of gravity (unit: m/s^2); total pressure ***difference*** value between inlet and outlet is ($p_{出} - p_{进}$) (unit: Pa).

($Z_{出} - Z_{进}$) is position shift of outlet and inlet (unit: m); volume traffic is Q (unit: m^3/s); w is rotation angular velocity (unit: rad/s); M is total moment of rotation axis (unit: N·m).

Evaluating the above 6 performances shows that in 1.0Q (design traffic), pump efficiency is the highest of 90%. Table 2 is test life and simulation lift in different traffic; Table 3 is test and simulation efficiency under the same traffic.

Table 1: *Total pressure values of inlet and outlet*

traffic (m^3/s)	**0.2 *Q***	**0.4 *Q***	**0.6 *Q***	**0.8 *Q***	**1.0 *Q***	**1.2 *Q***
Total pressure of inlet (Pa)	-13977.3	-23246.0	-34828.8	-42348.5	-36l74.8	-40066.1
Total pressure of outlet (Pa)	640349.7	616365.9	583691.7	557533.0	522798.9	461476.1

Table 2 and Table 3 suggest that simulation results are similar to test results. Specifically, delivery head declines while efficiency increases with traffic increasing; relative error is the smallest when design traffic is 1.0Q; and the more performance deviates from design performance, the bigger relative error is. The results produced by numerical simulation tally with the real situations. The comparison shows that efficiency in the simulation is higher than that in the test, for mechanical friction loss of the bearing's stuffing box in the pump, disk friction loss due to the friction of the impeller and the liquid leaked, and volume loss caused by leak, are not taken into consideration in the simulation. After these factors are included in it, results will be more accurate.

Table 2: *The test and simulation lift under different traffic*

	traffic *Q* (m^3/s)					
***H*(m)**	**0.2 *Q***	**0.4 *Q***	**0.6 *Q***	**0.8 *Q***	**1.0 *Q***	**1.2 *Q***
test lift	64.8	64.2	62	60.5	56	48.5
simulation lift	67.14	66.09	63.64	61.75	56.98	51.54
relative error	3.61%	2.946%	2.65%	2.07%	1.75%	6.27%

Table 3: *The test and simulation efficiency under different traffic*

	traffic *Q*(m^3/s)					
η **(%)**	**0.2*Q***	**0.4*Q***	**0.6*Q***	**0.8*Q***	**1.0*Q***	**1.2*Q***
test efficiency	26	50	68	82	88	83
simulation efficiency	27.3	51.6	69.7	83.9	90	85.6
relative error	5.00%	3.20%	2.50%	2.32%	2.27%	3.13%

6. Conclusions

This paper adopts Pro/Engineer software to built three-dimensional model of the 1200s－56 large-scale double-suction centrifugal pump which most often used in Jingtaichuan Pumping Irrigation Project in Gansu Province, ANSYS ICEM CFD is used to divide the mesh and uses k-ε turbulence model and wall function to analyze the three dimensional flow in the impeller of the pump. With water as medium, to simulate the flow field inside the pump in several conditions, Analyzed by simulation results the following conclusions can be drawn:

(1) We can obtain H-Q, η-Q data sheet of the pump from simulation, the research shows that the simulation results tally with the test results, which means when the design performance is 1.0Q, the relative error is the smallest. the result shows that simulation with software reflect flowing discipline inside of the pump to some extent.

(2) Explore the flow mechanism inside of the 1200s-56 large-scale double-suction centrifugal pump. During the working states, pressure of the impeller was gradually increased from entrance to exit. Besides, at the same radius, pressure of front side of vane was higher than the other side, It is mainly because the special construction of the pump which used half-spiral suction chamber.

(3) Simulating the double-suction centrifugal pump by Fluent can produce accurate flow field in it, which can contribute to modifying the large double-suction pump. CFD can select the best plan for the design and the model test of the pump without the comparison of different plans. Therefore, the numerical simulation test can greatly reduce the research and development

circle of the pump, and enhance its design quality and technology.

Acknowledgements

This work was financially supported by the “Natural Science Foundation of China (51279064; 31360204)”, the “Supported by Program for Science & Technology Innovation Talents in Universities of Henan Province (14HASTIT047)” and the “Supported by Program for Innovative Research Team (in Science and Technology) in University of Henan Province (14IRTSTHN028)”.

References

[1] Z. Yao, K. M. Chen. “Review on the Commercial CFD Softwares”, Journal of University of Shanghai for Science and Technology, Issue 2., pp.137~144, 2002.

[2] B. L. Cui, Y. G. Lin. “Numerical Simulation of Flow in Centrifugal Pump with Complex Impeller”, Journal of Thermal Science, Issue 1., pp.47~52, 2011.

[3] M. G. Tan, S. Q. Yuan, H. L. Liu et al. “Numerical Research on Performance Prediction for Centrifugal Pumps”, Chinese Journal of Mechanical Engineering, Issue 1., pp.21~26, 2010..

[4] Y. L. Zhang, Y. Li, B. L. Cui, et al. “Numerical Simulation and Analysis of Solid-liquid Two-phase Flow in Centrifugal Pump”, Chinese Journal of Mechanical Engineering, Issue 1., pp.53~60, 2013.

[5] L. Zhou, W. D. Shi, W. G. Lu et al. “Numerical Simulation and Experiment on Deep-well Centrifugal Pump”, Transactions of the Chinese Society for Agricultural Machinery, Issue 3., pp.69~73, 2011.

[6] C. G. Li, F. J. Wang, J. Z. Xu. et al. “Pressure fluctuation of a two-stage double-suction centrifugal pump”, Transactions of the Chinese Society for Agricultural Machinery, Volume 42, Issue 7., pp.41~49, 2011.

[7] J. González, E. Blanco, C. Santolaria, et al. “Unsteady flow structure on a centrifugal pump: experimental and numerical approaches”, Proceedings of the ASME Fluids Engineering Division Summer Meeting, Montreal, Canada, 761～768, 2002.

[8] P. C. Guo, X. Q. Luo, S. Z. Liu. “Optimized Design of the Centrifugal Pump Impeller based on Numerical Aanalysis of 3D Turbulent Flow”, Chinese Journal of Mechanical Engineering, Issue 4., pp.181~184, 2004.

[9] Y. C. Zhang, B. H. Luo, W. Q. Wu. “Research on Energy Saving Technology of the Double-suction Centrifugal Pump based on the CFD Technology”, Journal of Hydroelectric Engineering, Issue 4, pp.93~96, 2009.

[10] H. G. Sung, S. J. Kim, H. W. Yeom. “On the Assessment of Compressibility Effects of Two-Equation Turbulence Models for Supersonic Transition Flow with Flow Separation”, International Journal of Aeronautical and Space Sciences, Volume 14, Issue 4., pp.387~397, 2014.

[11] H. L. Liu, D. X. Liu, Y. Wang, et al. “Application of modified k-ω model to predicting cavitating flow in centrifugal pump”, Water Science and Engineering, Issue 3., pp.331~339, 2013.

[12] X. Y. Liu, T. X. Zhang, X. B. Dai, et al. “Pro/E-based design of infrared window of image-spectrum integrated instrument”, OPTIK, Volume 125, Issue 12., pp.2731~2733, 2014.

[13] K. Amail, M. Hugo, D. C. Alain. “CFD approach for modeling high and low combustion in a natural draft residential wood log stove”, International Journal of Heat and Technology, Volume 33, Issue 1., pp.33~38, 2015

[14] M. M. Seyed. “CFD modeling of nature convection in right-angled triangular enclosures”, International Journal of Heat and Technology, Volume 34, Issue 3., pp.503~506, 2016.

[15] W. Q. Tao. Numerical heat transfer. Xi 'an: xi 'an jiaotong university press, 1988.

[16] S. E. Rafiee, M. M. Sadeghiazad. “Three-Dimensional CFD simulation of fluid flow inside a Vortex tube on basis of an experimental model- the optimization of vortex chamber radius” International Journal of Heat and Technology, Volume 34, Issue 2., pp.236~244, 2016.

[17] Z. R. Liu, Z. C. Song, F. Liu, et al. “Numerical Simulation of Internal Flow Field in Centrifugal Water Pump Based on CFDesign”, Coal Mine Machinery, Issue 1., pp.57-59, 2009.

[18] J. J. Yang, D. W. Dong, Z. W. Meng, Y. H. Yang, Y. Wang. “Different Type of Flow Field and Engine Performance of The Vortex Throttle”, International Journal of Heat and Technology, Volume 34, Issue 2., pp.319~324, 2016.