Journal of Applied Sciences

ISSN 1812-5654

Journal of Applied Sciences 13 (17): 3416-3420, 2013
ISSN 1812-5654 / DOI: 10.3923/jas.2013.3416.3420

Numerical Analysis of a Prototype Centrifugal Pump Delivering Solid-liquid Two-phase Flow

Tang Hua, Li Yi and Zhang Yu-Liang
The Zhejiang Key Lab of Fluid Transmission Technology, Zhejiang Sci-Tech University,
Hangzhou, 310018, China

Abstract: In order to study solid-liquid two-phase turbulent flow, the CFD software FLUENT was used to simulate solid-liquid two-phase flow in a centrifugal pump based on the SIMPLEC algorithm and the Mixture model. By calculating, the distribution of solid phase particles by the influence of different particle diameters and different density were obtained and pump external characteristics were also predicted. The results indicated that increasing particle diameter and density will cause the reduction of pump head and efficiency. For the effect of inertia force, particles with large value of diameter and density are easy to move towards shroud and suction surface, which will aggravate impeller wear.

Key words: Solid-liquid two-phase, impeller, external characteristics, numerical simulation

INTRODUCTION

Centrifugal pump are widely used in the field of agricultural irrigation, urban water supply, boiler feed water, mining, oil and chemical industry (Li and Wang, 2006). Pump delivery medium contain solid particles for the limitation of work conditions. Therefore, it's very important significance to study influence of solid phase on centrifugal pump inner flow and external characteristics.

Impeller is the most critical flow components and its internal flow is a very complex three-dimensional flow. There is flow separation, secondary flow and other flow phenomena when pump under off-design working conditions (Medvitz *et al*., 2002). Many scholars have conducted in-depth study about impeller design (Cao *et al*., 2005; Zangeneh, 2008; Lu *et al*., 2002). Liu *et al.* (2010) analyze the impact of solid particles properties and other external factors on the distribution of particles. Chen *et al.* (2006), Zhao *et al.* (2008) and Liu *et al.* (2008) analyze solid-liquid flow pattern in order to solve problems of pump wear. Wang *et al.* (2009) obtained different sizes of the particles trajectory and time distribution in a centrifugal pump. By using the processing idea of filter function in Large Eddy simulation, (Wu *et al*., 2001) derived two-phase flow motion equations of large-eddy simulation.

From the nature departure of solid phase particles, this article studied particle motion and distribution in the impeller channel. Impeller flow pattern and these factors influence on the pump external characteristics were obtained by calculating in order to optimize design, reduce wear and extend service life of high specific speed centrifugal pump.

NUMERICAL MODEL AND METHOD

Geometric and multiphase flow model: Centrifugal pump main design parameters are shown in Table 1.
In this study, the Mixture model is used to simulate centrifugal pump two-phase flow.

Meshing and grids: The computational domain is divided into four parts: impeller rotating areas, volute static areas, suction area and volute extended area. The impeller rotation areas and volute static areas are meshed by tetrahedral unstructured grid but suction and volute extended areas are meshed by structured grid. The total number of computational grids are 1.21 million.

Boundary condition: For incompressible flow, inlet boundary is set to velocity inlet and assume that there is no tangential velocity and radial velocity.

Outlet boundary is set to free outflow because exit area is away from the recirculation zone and the outlet boundary conditions is the second type of boundary conditions. We have:

Table 1: Centrifugal pump main parameters

Parameters	Flow Q $m^{-3}\cdot h^{-1}$	Speed n (rpm)	Specific speed n_s	Inlet diameter D_1 (mm)	Outlet angle β_2/(°)	Outlet width b_2 (mm)	Impeller diameter D (mm)	No. of blades Z
Value	100	2900	196	89	23.5	25	140	6

Corresponding Author: Li Yi, The Zhejiang Key Lab of Fluid Transmission Technology, Zhejiang Sci-Tech University, Hangzhou, 310018, China

$$\frac{\partial \phi}{\partial n} = 0 \qquad (1)$$

In the equation ϕ-the parameters on the outlet boundary; n-normal direction on the outlet boundary.

For the affect of fluid viscosity, the no-slip boundary conditions are adopt at the wall. Because of the Reynolds number is very low in the near-wall region, turbulence model is no longer applicable. And we use the standard wall function method to deal with the near-wall flow.

RESULTS ANALYSIS

By using FLUENT software, different diameters, volume fractions and densities were calculated. Simulation scheme is shown in Table 2.

Performance prediction analysis: Performance prediction curve is shown in Fig. 1.

From Fig. 1, with the flow increasing, head drop and efficiency first increased and then decreased (efficiency reach the maximum value at optimum point). In Table 3, at design point (Q = 100 m^{-3} h), the relative error rate of head and efficiency are 2.4 and 5.7%, which are in the allowable error range. It prove the accuracy of this centrifugal pump model and numerical calculation method.

Solid phase parameters influence on pump external characteristics: Figure 2 to 4 analysis influence of different particle diameter, volume fraction and density on pump external characteristics.

Figure 2 shows that, as particle diameter increase, the efficiency and head of pump showed a decreasing trend but shaft power is almost constant. In Fig. 3, influence of particle volume fraction on pump head, efficiency and power is different. With volume fraction increasing, pump power increase and head and efficiency decrease because viscosity of the mixture liquid increase and pump friction loss increase. In Fig. 4, as density increase, pump shaft power increase but the head and efficiency show decreasing trend.

Influence of particle diameter: From Fig. 5,under the condition of optimum design point, particle density ρ = 1500 kg $m^{-1}$and solid volume fraction C_V = 10%, contours of particle distribution with different diameters in impeller channel. Particle concentration reduce from inlet to the outlet of the impeller. Despite the different particle diameter, the maximum of particle concentration is 0.1 in the impeller inlet. When d = 0.25 mm, particle distribute more uniformly than the condition of d = 0.05 mm.

Table 2: Scheme of simulation

Parameters	Diameter d (mm)	Volume fraction C_V (%)	Density ρ (kg•m^{-3})
Value	0.05, 0.10, 0.25	4, 10, 16	1100, 1500, 1900

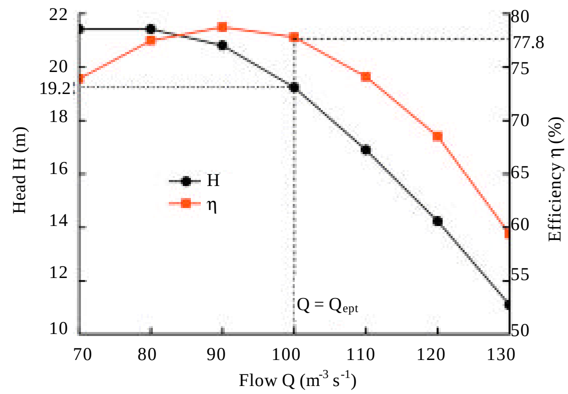

Fig. 1: Hydraulic performance curve

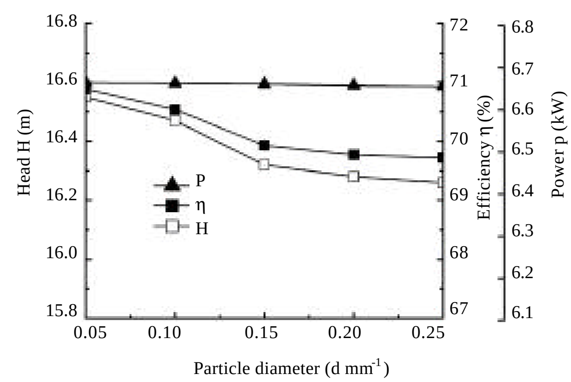

Fig. 2: Influence of particle diameter on external characteristics

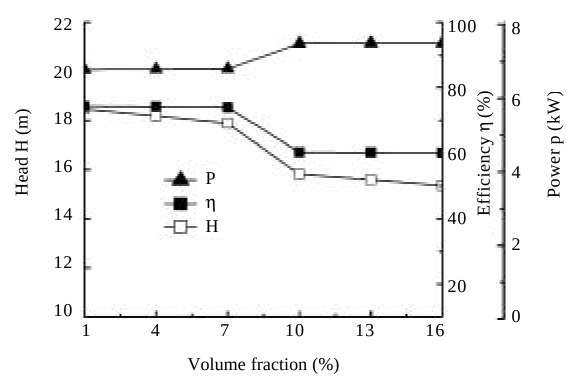

Fig. 3: Influence of volume fraction on external characteristics

Figure 6a, b shows the pressure (static pressure) distribution law on the blade by the influence of particle diameter. And abscissa is the relative length of the blade.

Table 3: Relative error rate at design point

Parameters	Hydraulic experiment	Numerical calculation	Relative error rate (%)
H/m	18.78	19.24	2.4
□/%	82.47	77.79	5.7

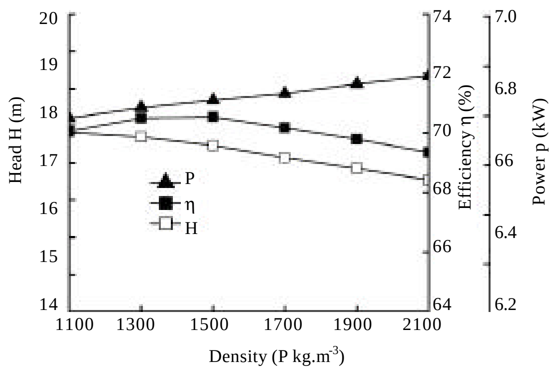

Fig. 4: Influence of density on external characteristics

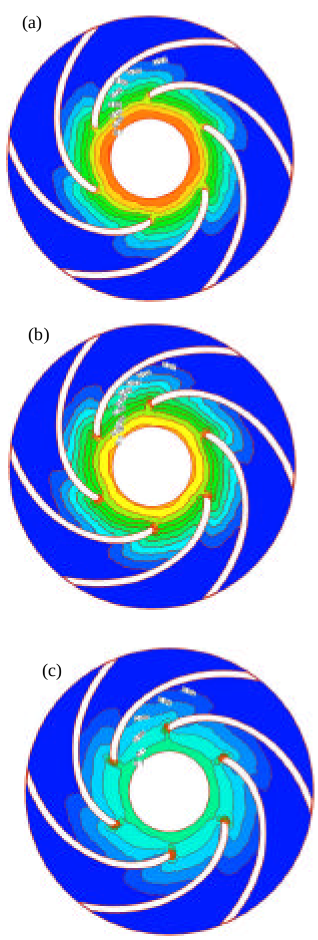

Fig. 5(a-c): Contour of particle distribution with different diameters in impeller channel (C_V = 10%), (a) d = 0.05 mm (b) d = 0.1 mm and (c) and d = 0.25 mm

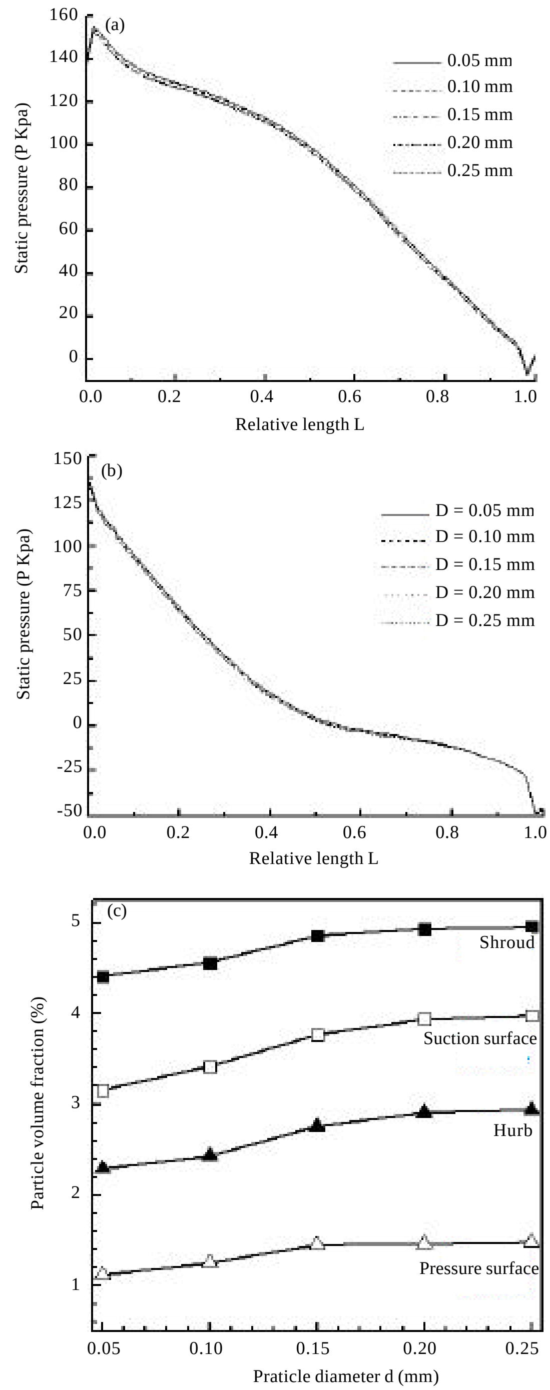

Fig. 6(a-c): Particle diameters influence, (a) Pressure surface, (b) Suction surface and (c) Particle volume fraction on the impeller

In the tubular direction of impeller channel, the rotation speed increases and the fluid acting capabilities continue to increase, which rise blade surface pressure and fully

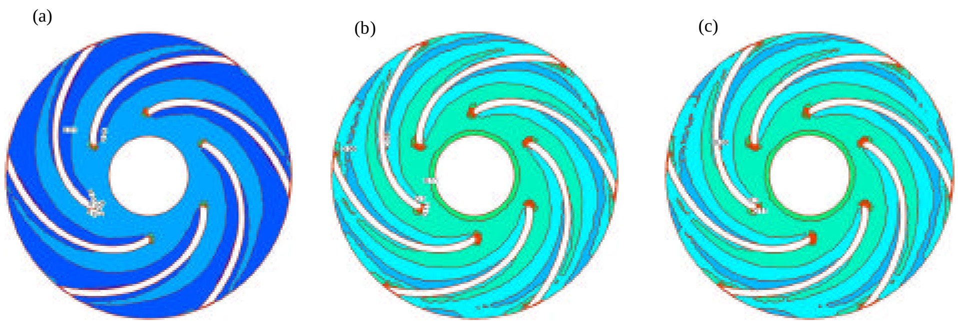

Fig. 7(a-c): Contour of particle distribution with different volume fractions in impeller channel d = 0.10 mm, (a) C_V = 4% (b) C_V=10% and (c) C_V = 16%

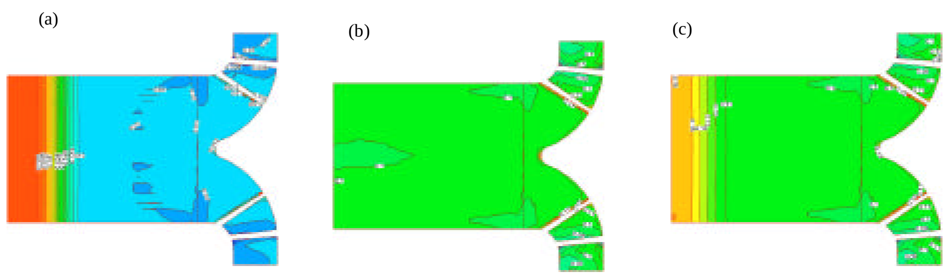

Fig. 8(a-c): Contour of particle distribution with different volume fractions from impeller inlet to outlet d = 0.10 mm, (a) C_V = 4% (b) C_V = 10% (c) C_V = 16%

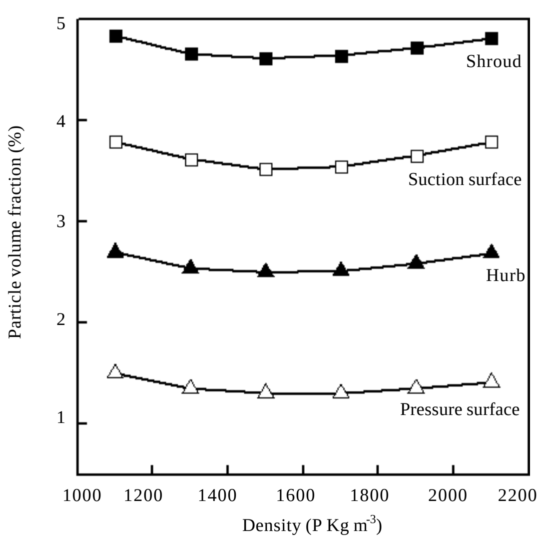

Fig. 9: particle volume fraction

reflects the acting effect of impeller to the fluid. From Fig. 6a, the maximum value of static pressure is 155.19 KPa and the minimum value is -7.83 KPa on the pressure surface. From Fig. 6b, the maximum value of static pressure is 136.43 Kpa and the minimum value is -48.56 KPa on the pressure surface. In Fig. 6c, as the particle diameter increases, particle distribute on the impeller component. Particles are mainly concentrated in the shroud and the suction surface.

Influence of solid phase volume fraction: Analysis of Fig. 7 and 8, the solid particles volume have a slight effect on particle distribution in impeller. As volume fractions increase, the particles have trend to move pressure surface. When C_V = 10%, particle distribute uniformly. In Fig. 8, when C_V = 4%, the value of particle concentration decrease from impeller inlet to outlet and the maximum value is 0.038. The minimum, which is 0.008, occurs at the impeller outlet. When C_V = 10%, however, the value of particle concentration have little change. The change range of particle concentration value is from 0.09 to 0.16 when C_V = 16%.

Influence of solid phase density: In Fig. 9, changes of particle density has little effect on the particles distribution. Particles are mainly concentrated in the shroud and the suction surface.

CONCLUSION

- Particle diameter, volume fraction and density have great influence on pump external characteristics. Pump lift and efficiency will reduce as these values increase
- With the increase of particle diameter and density, particles have strong trend to move towards the shroud and suction surface with the action of inertial force and their trajectories are obviously
- As the increase of solid phase volume fraction, particle concentration is gradually increased in the impeller channel. Particles friction with impeller at a certain speed could easily cause impeller localized wear

ACKNOWLEDGMENTS

This work is supported by the National Natural Science Foundation (51076144), National Science and Technology Support Project (2013BAF05B01).

REFERENCES

Cao, S.L., L. Liang and B.S. Zhu, 2005. Design method for impeller of high specific speed mix-flow pump. J. Jiangsu Uni., 26: 185-188.

Chen, C.C., C.M. Yang and M.T. Xiong, 2006. Numerical analysis on two-phase flow through impeller of low specific speed centrifugal pump. Drainage Irrigat. Machinery, 24: 1-3.

Li, Y.B. and Y. Wang, 2006. Improved design of non-overload super-low specific speed centrifugal pump. J. Drainage Irrigat. Machinery Engin., 24: 7-10.

Liu, J., H.Y. Xu, S. Tang and L. Lu, 2008. Numerical simulation of erosion and particle motion trajectory in centrifugal pump. Trans. Chinese Soc. Agric. Machinery, 30: 54-59.

Liu, J.R., Y.G. Xu and D.M. Wang, 2010. Numerical simulation of solid-liquid two-phase turbulent flow in impeller channel and characteristics-analysis. Trans. Chinese Soc. Agric. Machinery, 41: 86-90.

Lu, J.L., G. Xi and D.T. Qi, 2002. Research on the inverse design method for 3-D blades in centrifugal pumps. J. Engin. Thermo Phys., 23: 61-64.

Medvitz, R.B., L.L. Pauley, R.F. Kunz, D.A. Boger, J.W. Lindau and A.M. Yocum, 2002. Performance analysis of cavitating flow in centrifugal pumps using multiphase CFD. J. Fluids Engin., 12: 377-283.

Wang, Y., X.M. Xu and X. Zhang, 2009. Numerical simulation for the motion of dilute solid particles in centrifugal pump. Fluid Machinery, 37: 24-31.

Wu, Y., L. Ge, N. Chen, 2001. Large eddy simulation of siltliquid twophase flow through a centrifugal pump impeller. J. Tsinghua Univ.: Sci. Tech., 41: 93-96.

Zangeneh, M., 2008. A compressible three-dimensional design method for radial and mixed flow turbo machinery blades. Intl. J. Num. Meth. Fluids, 13: 599-624.

Zhao, B.J., S.Q. Yuan and H.L. Liu, 2008. Simulation of solid-liquid two-phase turbulent flow in double-channel pump based on mixture model. Trans. Chinese Soc. Agric. Engin., 24: 7-12.