Proceedings of the ASME 2010 3rd Joint US-European Fluids Engineering Summer Meeting and 8th International Conference on Nanochannels, Microchannels, and Minichannels
FEDSM-ICNMM2010
August 1-5, 2010, Montreal, Canada

FEDSM-ICNMM2010-30198

Cavitation Performance Study Based On Numerical Simulation of Magnetic Driving Pump

Fanyu Kong
Fluid Machinery Research Center, Jiangsu University, China

Xiaokai Shen
Fluid Machinery Research Center, Jiangsu University, China

Xuefeng Zhang
Fluid Machinery Research Center, Jiangsu University, China

Kuanrong Xue
Hangzhou Dalu Industry co. Ltd, China

Shuiqing Zhou
Fluid Machinery Research Center, Jiangsu University, China

Wengti Wang
Fluid Machinery Research Center, Jiangsu University, China

ABSTRACT

A new magnetic driving pump with low-NPSHR had been developed to ensure leakage-free for transporting fluid. In hydraulic design, several common measures, according to the equation of cavitation ,were applied to lower down the NPSHR of the pump. Moreover, a variable-pitch inducer whose pitch increases gradually was installed at the inlet of impeller. Cavitation performance of the magnetic driving pump and the reasons why the NPSHR was reduced by installing a variable-pitch inducer were analyzed according to the simulation predicted flow field at the pump inlet and inducer. The test of cavitation performance showed that the magnetic driving pump had the excellent cavitation performance, and met the design requirements in flow, head and vibration. The magnetic driving pump presented the reliability after a long period of operation in a petrochemical corporation. Therefore, magnetic driving pumps with low-NPSHR would have a great future in petrochemical, shipping and other important fields.

Keywords: Magnetic driving pump; low-NPSHR; Impeller; Inducer; Numerical simulation.

INTRODUCTION

Since the cobalt magnetic transmission was first developed in Germany in 1976, many companies had developed series of magnetic driving pumps. In 1983, a new permanent magnetic material, Nd-Fe-B, came out, which promoted the development of magnetic driving pump. Currently, the power of magnetic driving pump can be up to 350kW,the fluid temperature can attain 450℃ and working pressure can be 4.5MPa[1-3].In China, the investigation on magnetic driving pump started in 1970s, and a big disparity ,compared to developed countries, really exists in the large power magnetic driving pump. Jiangsu University Fluid Machinery Research Center had taken great achievements in the design of pump and variable-pitch inducer[4,5], which contribute to the development of magnetic driving pump in China. And the magnetic driving pump with power of 200KW had appeared in China. China is rich in permanent magnetic materials, which gives a brilliant prospect for developing excellent performance magnetic driving pump.

The centrifugal pumps used in petrochemical, ship, power station and aerospace to pump inflammable, explosive and toxic fluid have the requirements of non-leakage, low- NPSHA and low-NPSHR. Currently, the investigations on magnetic driving pump with low-NPSHR are really rare. This paper, based on numerical simulation, presents the cavitation performance and distributions of flow field in impeller and inducer. So the paper is helpful to develop low-NPSHR magnetic driving pump.

HYDRAULIC DESIGN OF LOW-NPSHR PUMP

The NPSHA of is unchangeable. Lowering NPSHR is only way to improve the cavitation performance[6]. So the pump impeller with good cavitation performance and a variable-pitch inducer installed at front of impeller are necessary in the design[7]. The structure is showed in Fig.1.

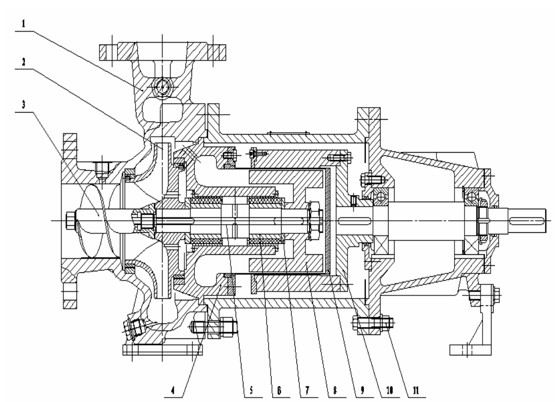


Fig.1 Structure of the low-NPSHR magnetic driving pump
1.Pump body 2.Impeller 3.Inducer 4.Pump cover 5.Shaft 6.Guide bearing 7.Thrust washer 8.Inner coupling 9.Containment shell 10.Outer coupling 11.bearing bracket parts

Low-NPSHR Impeller

Cavitation equation is expressed in Eq(1):

$$\text{NPSHR}=\frac{v_0^2}{2g}+\lambda\frac{w_0^2}{2g} \tag{1}$$

Where v_0—— the average absolute velocity of fluid in the front of blade;

w_0—— the average relative velocity of fluid in the front of blade;

λ—— pressure-drop coefficient of fluid in blade inlet.

According to the Eq(1), NPSHR could be reduced by decreasing v_0, w_0, and λ.

(1) The diameter of impeller inlet D_j should be increased , while the diameter of hub d_h should be reduced, as showed in Fig.2.

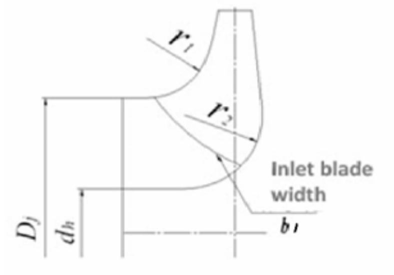


Fig.2 Impeller parameters

The area of pass flow is that impeller inlet area subtracts hub sectional area. The diameter of area of pass flow D_0 is expressed in Eq(2):

$$\frac{D_0^{\,2}\pi}{4}=\frac{(D_j^{\,2}-d_h^{\,2})\pi}{4} \tag{2}$$

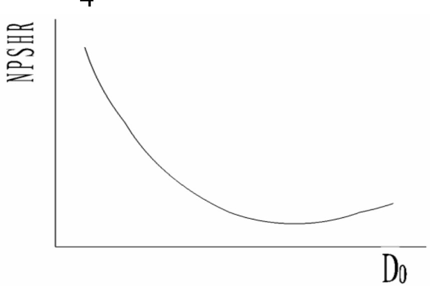


Fig.3 The curves of NPSHR and D_0

Fig.3 shows the relation between NPSHR and D_0 .It shows that NPSHR could be reduced by increasing D_0 . However, the NPSHR would not be decreased any more after the D_0 reaches some value.

(2) The inlet edge of blade could be enlaged in suction direction.

The inlet edge of blade in suction direction made the blade influence on flow in advance, which could increase the surface of blade, decrease pressure difference between the pressure surface and suction surface and reduce the radius of impeller inlet.

(3) Curvature radius of front shroud r_L should be increased.

The increment of curvature radius of front shroud is helpful to improve velocity distribution, avoid flow separation and decrease pressure-drop coefficient.

(4)The width of inlet edge of blade should be increased.

The area of inlet pass flow could be increased by enlarging the width of inlet edge of blade.

(5)Inlet blade angle should be increased.

Blades with larger positive incidence angle could decrease curvature of blade inlet side and enlarge the area of inlet pass flow, reduce v_0 and w_0 , and avoid negative incidence in the condition of large flow.

(6)Other measures

Less blades and thinner blade inlet could enlarge the area of flow passage, which can weak the resistance of flow and reduce friction loss.

Variable-pitch Inducer

NPSHR couldn't be reduced largely only through a reasonable impeller design. There is another way to install a inducer in the front of impeller. The energy produced by inducer was worked on pump impeller to raise the pressure at impeller inlet, and ensure no cavitation could happen. The inducer itself must have excellent cavitation performance to ensure the pump run without cavitation happens. The essential of the way is that the NPSHR is reduced[8,9]. Therefore, the inducer plays an important role on the design of the low-NPSHR pump[10].

The constant-pitch inducer has some defects in hydraulic performance. Because its pitch is invariable from inlet to outlet, which means that flow area and blade angle can only be constant. The actual flow and energy transformation of fluid showed the constant-pitch inducer was inappropriate.

So, in order to meet requirements about cavitation of the inducer and pump inlet pressure, a variable-pitch inducer was applied[11].

For a variable-pitch inducer, the small blade inlet angle canlead to a small inlet flow coefficient, and the big blade outlet angle can product enough head. It met the qualification about cavitation of inducer and impeller inlet pressure.

NUMMERICAL STUDY

Parameters

$Q=200m^3/h$, H=80m, n=2950r/min, NPSHA=2.5m.

Generally, NPSHR of a pump with these parameters is 5.5m, and it does not meet the requirement of cavitation. Thus, low-NPSHR design for impeller and a variable-pitch inducer were considered in pump design. The inducer head must be bigger than 3m. According to the parameters above, the parameters of impeller are showed on Tab.1,and the parameters of variable-pitch inducer are showed on Tab.2.

Tab.1 Parameters of impeller (mm)

Parameter	Inlet diameter (D_j)	Hub diameter (d_h)	Blade inlet width (b_1)	Front shroud radius (r_1)	Back shroud radius (r_2)
Value	160	60	65	34	42

Tab.2 parameters of the variable-pitch inducer

Parameter	Inlet flow coefficient φ_{t1}（°）	Flange diameter D_i（*mm*）	Hub diameter D_h（*mm*）	Flange inlet install angle βt_1（°）
Value	0.15	159	40	9
Paramete	Flange outlet install angleβ_{t2}（°）	Axial length H(*mm*)	Inducer blade number Z	Wrap angle φ（°）
Value	24	83	2	246

Model

Based on the parameters above, the models of impeller, inducer and volute were built by 3-D software (Pro/E). The lengths of inlet and outlet were taken as 2 to 4 times as their diameters to simulate flow field accurately. The models are showed in Fig.4

The CFD software in this simulation is FLUENT. FLUENT software was developed by FLUENT company in 1983, which is one of the best CFD software and is most widely used in CHINA. FLUENT can offer many kinds of mesh property and users can choose different kinds of mesh, including triangle mesh, quadrilateral mesh, tetrahedral mesh, hexahedral mesh , etc.

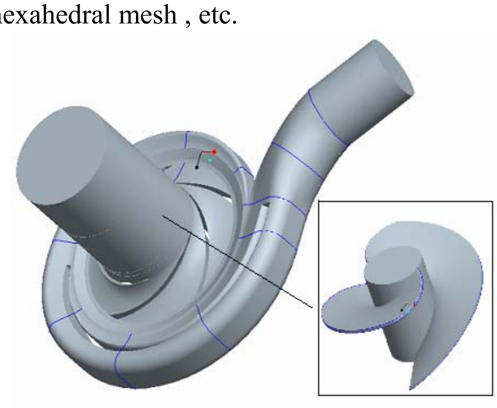


Fig.4 Models of impeller, inducer and volute

Equation And Turbulent Model

The flow in the pump is supposed to be 3-D and incompressible, three-dimensional turbulent flow. It meets continuity equation and momentum equation[12].

Continuity equation is expressed in Eq.3：

$$\frac{\partial \rho}{\partial t}+\frac{\partial}{\partial x_i}(\rho u_i)=0 \tag{3}$$

Momentum equation is expressed in Eq.4：

$$\frac{\partial}{\partial t}(\rho u_i)+\frac{\partial}{\partial x_j}(\rho u_i u_j)=-\frac{\partial p}{\partial x_i}+\frac{\partial}{\partial x_j}(\mu\frac{\partial u_i}{\partial x_j})+S_i \tag{4}$$

Where ρ— fluid density, Kg/m^3;
u— velocity, m/s;
p— average pressure, Pa;
t — time, s;
x— space coordinate;
μ—dynamic viscosity, Pa.s;
S—source;
i, j —coordinate axis direction component.

Standard *k-ε* double equation turbulence model was applied in simulation. General transportation equations of turbulent kinetic energy(k) and turbulent dissipation rate(ε) were adopted to combine with Eq.2 and Eq.3 to form Eq.5[13.]

$$\begin{cases}\frac{\partial}{\partial t}(\rho k)+\frac{\partial}{\partial x_i}(\rho k u_i)=\frac{\partial}{\partial x_j}[(\mu+\frac{\mu_t}{\sigma_k})\frac{\partial k}{\partial x_j}]+G_k-\rho\varepsilon+S_k \\ \frac{\partial}{\partial t}(\rho \varepsilon)+\frac{\partial}{\partial x_i}(\rho \varepsilon u_i)=\frac{\partial}{\partial x_j}[(\mu+\frac{\mu_t}{\sigma_\varepsilon})\frac{\partial \varepsilon}{\partial x_j}]+C_{1\varepsilon}\frac{\varepsilon}{k}G_k-C_{2\varepsilon}\rho\frac{\varepsilon^2}{k}+S_\varepsilon\end{cases} \tag{5}$$

Where

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

Where μt—turbulent viscosity coefficient;
Gk—turbulent kinetic energy;
σ_k—Prandtl number of k equation,σk =1.0;
σ_ε— Prandtl number of εequation, σε=1.3;
S_k—viscous stress;
S_ε—turbulent kinetic stress.

Constant number:

C_μ=0.09；$C_{1\varepsilon}$=1.44；$C_{2\varepsilon}$=1.92.

SIMPLE calculation method was applied as solution method to solve pressure and velocity; a under-relaxation factors was used for accelerating convergence; velocity inlet was used as pump inlet boundary condition; outflow was used as outlet boundary condition; surfaces of impeller blades, inducer blades, front shroud and back shroud, and volute case were considered as non-slip firm wall[14].

Results and analysis

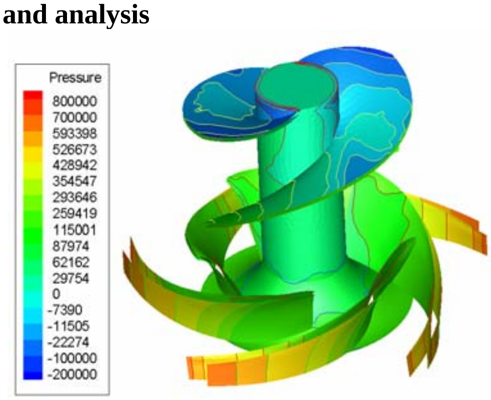


Fig.5 Static pressure distribution

Fig.5 shows that the flow pressure increases gradually from inducer inlet to impeller outlet, but the pressure distribution is even in the most areas along flowing direction; the pressure of fluid in impeller inlet is higher than that in inducer inlet, which manifests that the inducer in front of impeller increases the static pressure of flow before it entrances into the impeller, so as to avoid cavitation in impeller.

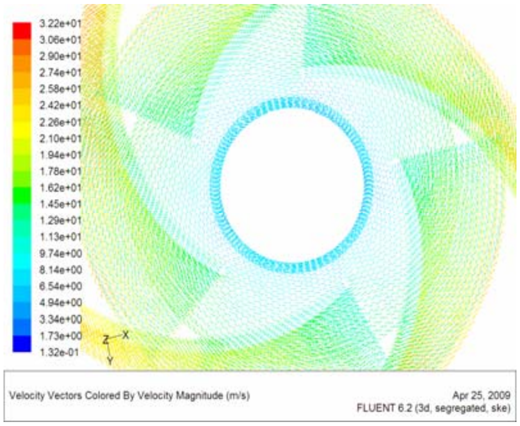
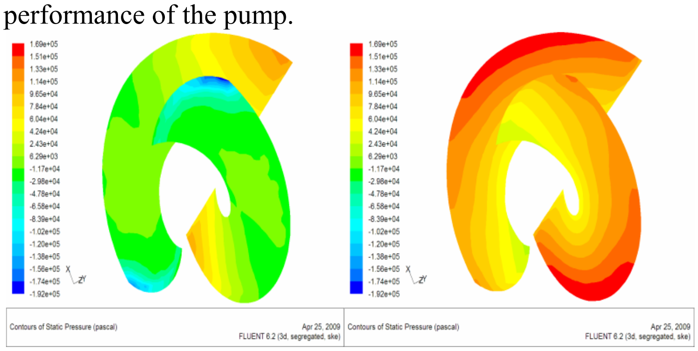


Fig.6 Velocity distribution of blade and hub surface

Fig.6 shows that the blades designed with low-NPSHR method make the flow velocity change smoothly and more energy is exchanged from velocity to pressure, which makes the flow pressure be higher to improve the cavitation performance of the pump.



(a)Pressure on suction face (b)Pressure on working face

Fig.7 Static pressure distributions on inducer blade surfaces

Fig.7 shows that pressure on working face is higher than that on suction face, which is conform to the functinon of inducer. The flow pressure increases gradually from the front edge of inducer to outlet. Especially on front half part of inducer blades, flow pressure increases greatly. The pressure of fluid on both of suction surface and working face is increased effectively along flowing direction. In inducer outlet, the pressure distribution at radial and axial direction is even, which is benefit for fluid to entrance into the impeller.

The pressure on inlet cavetto of suction face is very low, where bubbles generate easily. The relative velocity reaches maximum here, however, the blades have not worked on the flow yet. At this moment, most of static pressure energy is exchanged into velocity energy, so the pressure here is very low. If cavitation occurs in inducer, bubbles would generate at inducer flange. The bubbles would rupture while moving along axial direction, so this cannot block at the whole conduit. Thus, the inducer can run with a little cavitation, and do not have a large influence on performance.

EXPERIMENT

The pump was tested on a standard experiment table, the curves of performance and cavitation performance are showed in Fig.8.

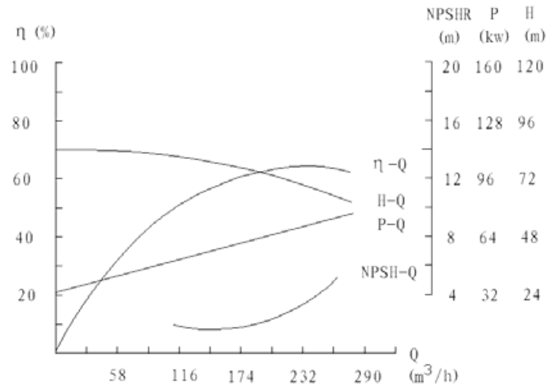


Fig.8 Performance curve of pump

Fig.8 shows that the flow and head of pump meet the design requirements. At design point, NPSHR is 2.27m and NPSHA is 2.5m, so the pump meets the requirement of cavitation performance. Besides, the pump ran steady in operating range, and vibration severity was range of 0.17-0.86mm/s, meeting the qualification of A class. The pump presented the excellent reliability after a period time of operation in a petrochemical corporation.

CONCLUSION

(1) It's reasonable to design the impeller with low inlet velocity and fix an variable-pitch inducer in front of impeller to improve cavitation performance.

(2) The inducer can improve the cavitation of pump. The pressure increased largely in inducer inlet, while the distributions of pressure and velocity in impeller inlet were homogeneous.

(3) The low-NPSHR magnetic driving pumphas the characteristics of non-leakage , excellent cavitation performance and stead operation to pump special fluid, meeting special requirements..

REFERENCES

[1] Semenov, Y, Fujii, A. , Tsujimoto, Y. 2004, "Rotating Choke in Cavitating Turbopump Inducer". *Transactions of the ASME, Journal of Fluids Engineering*, v126: 87～93.

[2] Majidi, K. 2005, "Numerical Study of Unsteady Flow in a Centrifugal Pump". *Journal of Turbomachinery*, 127: 363～371.

[3] "Goulds introduces Dry Guard bearings for magnetic-drive pumps". *World Pumps*, Volume 2006, Issue 477, June 2006, 8.

[4] Kong Fan-yu, Liu Jian-rui. 2005, "Develop Low Power High Speed Magnetic Driving Pump".*Pump*

Technology, . 3, 12-16.

[5] SUN-Jian, KONG Fan-yu, et, al.. 2007, "Design Calculation on Variable-pitch Inducers". *Transactions of the Chinese Society for Agricultural Machinery*, 38(2):102-110.

[6] ZHU Zu-chao.1999, "Theoretical Analysis and Engineeting Implementation on Achieving Excellent Suction Performance of High Speed Centrifugal Pumps". *Chinese Journal of Mechanical Engineering*, 35(6):85-88.

[7] Hong SS, Kim JS, Choi CH, Kim J. 2006, "Effect of tip clearance on the cavitation performance of a turbopump inducer". Journal of Propulsion and Power, 22(1), 174-179.

[8] KONG Fan-yu, Lü Yi, et, al.. 2007, "Numerical Simulation on Flow Field of Variable-pitch Inducers". *Transactions of the Chinese Society for Agricultural Machinery*, 38(10):45-48.

[9] Yao-jun LI and Fu-jun WANG.. 2007, "Numerical Investigation of Performance of an Axial-Flow Pump With Inducer". *Journal of Hydrodynamics, Ser. B*, 19(6), 705-711.

[10] JI Feng-lai, WU Yu-zheng. 2007, "Analysis and Design of Variable Screw-pitch Inducer with Excellent Performance". *Process Equipment & Piping*, 44(3):35-37.

[11] HUANG Jian-de, GU Chuan-gang. 2000, "The Effect of Inducer Shape on the Alternate Blade Cavitation". *Journal of Aerospace Power*, 15(2):142-146.

[12] OKITA K., UGAJIN H., MATSUMOTO Y.. 2009, "Numerical analysis of the influence of the tip clearance flow on the unsteady cavitating flow in a three dimensional". *Journal of Hydrodynamics*, 21(1):34-40.

[13] WANG Fu-jun. 2005, "Application of CFD to turbulent flow analysis and performance prediction in hydraulic machinery". *Journal of China Agricultural University*, 10(4):75-80

[14] YI Tong-xun, CHENG Fang-zhen, et, al.. 2000, "Numerical investigation of the flow field inside the liquid hydrogen inducer". *Journal of Propulsion Technology*, 21(4):9-11.

.