Original Paper (Invited)

Investigation on the Flow Field Upstream of a Centrifugal Pump Impeller

Yao Zhang¹, Xianwu Luo¹, Yunchi Yi², Baotang Zhuang¹, Hongyuan Xu¹

¹State Key Laboratory of Hydrosience & Engineering, Tsinghua University
Beijing, 100084, China, yao-zhang06@mails.tsinghua.edu.cn, luoxw@tsinghua.edu.cn

²Qinghe school, Beijing Vocational College of Agriculture
Beijing, 102208, China

Abstract

The flow upstream of a centrifugal pump impeller has been investigated by both experimental test and numerical simulation. For experimental study, the flow field at four sections in the pump suction is measured by using PIV method. For calculation, the three dimensional turbulent flow for the full flow passage of the pump is simulated based on RANS equations combined with RNG k - ε turbulence model. From those results, it is noted that at both design load and quarter load condition, the pre-swirl flow whose direction is the same as the impeller rotation exists at all four sections in suction pipe of the pump, and at each section, the pre-swirl velocity becomes obviously larger at higher rotational speed. It is also indicated that at quarter load condition, the low pressure region at suction surface of the vane is large because of the unfavorable flow upstream of the pump impeller.

Keywords: centrifugal pump, flow upstream impeller inlet, PIV, numerical simulation.

1. Introduction

It is known that the flow inside a centrifugal pump is extremely complicated and highly turbulent, caused by streamline curvatures, flow separations, rotor-stator interaction and turbulence effects ^[1]. Thus, the internal flow is crucial and need to be investigated in detail.

Besides, the flow upstream of an impeller is another factor which can affect hydraulic performance as well as cavitation performance of centrifugal pumps and can not be neglected. The pre-rotation and vortex flow formed in the upstream flow field of a radial impeller may deteriorate the incoming flow condition for blades and thus lower the performance of pumps.

Kim ^[2] found that the performance of centrifugal compressors could be seriously affected by inlet flow distortions due to the unsatisfactory nature of the inlet configuration and the resulting inlet flow structure. Matsunuma ^[3] used LDV system to study the effect of the turbine nozzle secondary vortices on the flow field inside the rotor passage. It is proven that the nozzle passage vortex could induce large fluctuation of the rotor flow field. Engeda ^[4] designed three different inlet systems and investigated the influence of each inlet type on the compressor performance based on steady-state simulation. Yang ^[5] studied the influence of inlet recirculation on the flow structure of centrifugal impellers.

In order to reduce the pre-rotation flow in the intake pipe at small operating flow rates, Predin ^[6] adopted the method of add flow at the radial impeller entrance eye. The results show that the Q - H performance curve became more stable with additional flow.

In this paper, the measurement of the velocity distribution inside the suction pipe of a pump was conducted by using PIV (Particle Image Velocimetry) technique. Besides, steady simulations were conducted for the detail analysis of the flow field upstream of a centrifugal pump. The experimental and numerical results indicate that the pre-swirl vortices in the suction pipe can reduce the performance of the centrifugal pump.

2. Experimental Apparatus

2.1 Test Impeller and Test Sections

The detail structure of the test pump as well as the suction sections is shown in Fig.1. A centrifugal impeller is installed in a volute casing, whose front cover is connected with a suction pipe. In order to investigate the internal flow upstream of the impeller,

4 sections named as section 1-1', 2-2', 3-3', 4-4' are defined. It is noted that section 4-4' is right upstream of the impeller. The parameters for the impeller are shown in Table.1.

Fig.1 Section view of the test pump

Table 1 Specification of Impeller

Geometrical parameter		Values
Inlet diameter	$D_1(\text{mm})$	64.7
Outlet diameter	$D_2(\text{mm})$	100
Outlet width	$b_2(\text{mm})$	8.53
Blade outlet angle	$\beta_2(^{\circ})$	90
Blade number	z	12
Front shroud type	-	Semi-open

2.2 Test Rig

The experiments are carried out in the test rig shown in Fig.2^[7]. The test pump is driven by an electric motor. A tank with 0.06m^3 is set upstream of the test pump to feed and recollect water and also to separate bubbles. Two rotational speeds i.e. 500 and 1000 min^{-1} are chosen for tests. For the purpose of direct observation as well as PIV measurement, the suction pipe, impeller and volute casing were all made of transparent plexiglass.

2.3 Particle Image Velocimetry System

PIV is a technique that measures the instantaneous velocity field within an illuminated plane of the fluid field using light scattered or fluoresced from particles seeded into the fluid^[8].

Here, A CCD camera (PCO Sensicam $1.3\text{K} \times 1\text{K}$) is used for image acquisition. The laser and camera are synchronized by TSI software.

Four sections in the suction pipe illustrated in Fig.1 are chosen as the measuring planes. Two dimensional velocities are measured, while the axial velocity is not measured.

Fig.2 Schematic of test rig

3. Numerical Calculation Method

3.1 Governing Equations

Here, the commercial CFD software—ANSYS CFX 11.0 is used for the steady-state numerical simulation. The three-dimensional Reynolds stress-averaged Navier-Stokes equations are given as follows:

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{U}) = 0 \quad (1)$$

$$\frac{\partial}{\partial t}(\rho \mathbf{U}) + \nabla \cdot (\rho \mathbf{U} \mathbf{U}) = -\nabla p + \nabla \cdot [(\mu + \mu_t) \nabla \mathbf{U}] + \frac{1}{3} \nabla(\mu + \mu_t) \nabla \cdot \mathbf{U} \quad (2)$$

For the turbulence model, the RNG k - ε model is adopted since it has been proven to be superior for simulating turbulence separation flow.

$$\frac{\partial(\rho k)}{\partial t} + \nabla \cdot (\rho \mathbf{U} k) = \nabla \cdot \left[\left(\mu + \frac{\mu_t}{\sigma_k} \right) \nabla k \right] + P_k - \rho \varepsilon \quad (3)$$

$$\frac{\partial(\rho \varepsilon)}{\partial t} + \nabla \cdot (\rho \mathbf{U} \varepsilon) = \nabla \cdot \left[\left(\mu + \frac{\mu_t}{\sigma_\varepsilon} \right) \nabla \varepsilon \right] + \frac{\varepsilon}{k} [C_{\varepsilon 1} P_k - C_{\varepsilon 2} \rho \varepsilon] \quad (4)$$

where

$$\mu_t = C_\mu \rho \frac{k^2}{\varepsilon} \quad (5)$$

$$P_k = \mu_t \nabla \mathbf{U} \cdot (\nabla \mathbf{U} + \nabla \mathbf{U}^T) - \frac{2}{3} \nabla \cdot \mathbf{U} (3\mu_t \nabla \cdot \mathbf{U} + \rho k) + P_{kb} \quad (6)$$

$C_{\varepsilon 1}$, $C_{\varepsilon 2}$, C_μ , σ_k , σ_ε are constants, and their values in the RNG k - ε turbulence model are: $C_{\varepsilon 1} = 1.42 - \frac{\eta(1 - \frac{\eta}{\eta_0})}{(1 + \beta \eta^3)}$, $C_{\varepsilon 2} = 1.68$, $C_\mu = 0.085$, $\sigma_k = 0.7179$, and $\sigma_\varepsilon = 0.7179$. Note that $\eta_0 = 4.38$ and $\beta = 0.012$ (where $\eta = Sk/\varepsilon$, $S = (2\overline{D_{ij}D_{ij}})^{1/2}$).

3.2 Grid and Boundary Conditions

The simulation domain includes suction pipe, the impeller blade passages and volute casing as shown in Fig.3. The structured mesh is adopted for all computation part. In detail, the grid number for suction pipe shown in Fig.4 is 700,000. The grid numbers for impeller and volute casing are 800,000 and 600,000 respectively.

It is noted that the y^+ value for most of the solid surface is in the range of 30~100, which is acceptable for the present study.

The inflow adopted mass flow boundary condition. And the average static pressure boundary condition is imposed at the outlet surface. Besides, the interfaces between stationary and rotational zone are set as “frozen rotor”. The convergence criterion is set as residual of 10^{-4} .

Fig. 3 Computation model for the centrifugal pump

Fig. 4 Grid of the suction pipe

4. Results and Discussions

Two rotational speeds, 500 and 1000 min^{-1} are used. For each speed, two different flow rates at design operation condition $Q=Q_d$ and at a quarter-load condition $Q=0.25Q_d$ are tested experimentally and calculated numerically.

4.1 Flow Field at Design Load

The experimental and calculated absolute velocity results at four sections in the suction pipe at $Q=Q_d$ are shown in the Fig.5~6.

Because the shaft makes a shadow of the laser light at section 3-3' and 4-4', only part of the surface is recorded in the experimental results.

Fig. 5 Absolute Velocity contour at 500 min^{-1} (Q_d)

Fig. 6 Absolute Velocity contour at 1000 min^{-1} (Q_d)

By comparison between the calculated and PIV results, it is found that the velocity magnitude and the direction is approximately the same, which illustrates the validity of numerical strategy.

The results indicate that there exists pre-rotation in the suction pipe, whose direction is the same as impeller rotation at all sections. Besides, at each section, it is found that the pre-rotation velocity at 1000 min^{-1} is obviously larger than that at 500 min^{-1} .

This indicates that at higher rotational speed condition the pre rotation flow is more intense.

Also, at each rotational speed, e.g. 500 min^{-1} , the pre-swirl velocity gradually increases from section 1-1' to section 4-4'.

4.2 Flow Field at Quarter Load

The experimental results at $Q=0.25Q_d$ for 4 sections at two rotational speeds are shown in the Fig.7~8. The calculated results also agree well to the PIV results.

Fig. 7 Absolute Velocity contour at 500 min^{-1} ($0.25Q_d$)

Fig. 8 Absolute Velocity contour at 1000 min^{-1} ($0.25Q_d$)

Compare with the results at design load point, the pre-rotation flow also exists for all sections at this condition. Similarly, the direction of pre-rotation vortex is the same as the impeller rotation. Besides, the pre-rotation velocity at 1000 min^{-1} condition for

each section is much larger than that at 500 min^{-1} .

Moreover, the vortex flow at quarter load condition ($0.25Q_d$) seems stronger compared with the same condition (at equal rotating speed, at equal section) at design load (Q_d). It indicates that at low flow rate point, the internal flow in the suction pipe is highly turbulent, which can partly illustrates the stall at the impeller inlet.

4.3 Internal Flow Analysis

Fig.9~10 show the calculated averaged velocity at tangential, radial and axial direction and test results at tangential direction for four sections at $0.25Q_d$ and Q_d conditions when rotational speed equals to 500 (Fig.9) and 1000 min^{-1} (Fig.10) respectively.

The test results of tangential velocity shown in Fig.9 (a) and Fig.10 (a) further proven that the rotational speed at quarter load condition is larger than that at design load condition. Besides, with the increase of section radius, tangential velocity also increases monotonously. It means that the pre-swirl of flow is the most intensive at rim of the section.

The calculated tangential velocity shown in Fig.9 (b) and Fig.10 (b) has the same trend with the experimental result, which illustrates that the present numerical simulation to the pre-swirl flow in the suction pipe of pump is correct and applicable. The radial velocity shown in Fig.9~10(c) is very small compared with the velocity at two other directions. And the velocity distribution along radius is highly turbulent.

For the axial velocity shown in Fig.9~10(d), the velocity at design load condition is obviously larger than that at quarter load condition because of the influence of flow discharge. It also reveals that back flows exist at both rotational speeds.

Fig. 9 Averaged velocity distribution at 500 min^{-1}

Fig. 10 Averaged velocity distribution at 1000 min⁻¹

Fig.11 shows the pressure coefficient ($(p-p_0)/(0.5\rho U_2^2)$) on vane suction surface, where the reference pressure p_0 is the inlet pressure of the pump. Since the cavitation of pump always first occurs at suction surface of the blade near inlet side, the dimensionless pressure distribution shown in Fig.11 can partly illustrate the cavitation performance of the pump at different rotational speeds and flow discharges. From the calculated results, it is found that the low pressure region near impeller inlet at quarter load condition is obviously larger than that at the design load condition. This is mainly caused by the unfavorable flow at the upstream. Thus, it is necessary to improve the flow uniformity if cavitation performance of the pump is considered.

Fig. 11 Static pressure coefficient on vane suction surface

5. Conclusions

Based on those characteristics of upstream flow in the suction pipe, the follows can be concluded:

- (1) Whether at design load or quarter load condition, the pre-swirl flow whose direction is the same as the impeller rotation exists at all observed sections in suction pipe of the pump.
- (2) At each section, the pre-swirl velocity at 1000 min⁻¹ is obviously larger than that at 500 min⁻¹.
- (3) At quarter load condition, the low pressure region at suction surface of the vane is large because of the unfavorable upstream flow.

Acknowledgments

The research is supported by National Natural Science Foundation of China (No: 50976061), State Key Laboratory of Hydrosience & Engineering, Tsinghua University (No: 2010-ZY-4), and the Specialized Research Fund for the Doctoral Program of Higher Education of China (No.20090002110052).

Nomenclature

b	Vane width [mm]	Q	Pump flow discharge [m ³ /s]
D	Impeller diameter [mm]	U, u	Peripheral or tangential velocity (m/s)
n	Rotational speed [min ⁻¹]	β	Vane angle (°)
p	Static pressure (Pa)		
	Subscripts	r	radial direction
1	at the inlet of an impeller	u	tangential direction
2	at the exit of an impeller	z	axial direction

References

- [1] Feng J.J., Benra F.K., Dohmen H.J., 2009, "Time-resolved particle image velocimetry (PIV) measurements in a radial diffuser pump," Proceedings of the ASME 2009 Fluids Engineering Division Summer Meeting (FEDSM2009), Vail, Colorado USA, Aug.2-6, 2009, FEDSM2009-78297.
- [2] Kim Y., Engeda A., Aungier R., et al., "The influence of inlet flow distortion on the performance of a centrifugal compressor and the development of an improved inlet using numerical simulations," Proceedings of the Institution of Mechanical Engineers, Part A: Journal of Power and Energy, 215(3): 323-338.
- [3] Matsunuma T., 2007, "Effects of nozzle secondary vortices on unsteady hub-endwall flow of a turbine rotor," International Conference on Power Engineering-2007, Hangzhou, China, Oct. 23-27, pp. 331-337.
- [4] Engeda A., Kim Y., Aungier R., et al, 2003, "The Inlet flow structure of a centrifugal compressor stage and its influence on the compressor performance," Journal of Fluids Engineering, 125(5): 779-785.
- [5] Yang C., Chen S., Li D., 2010, "Inlet recirculation influence to the flow structure of centrifugal impeller," Chinese Journal of Mechanical Engineering, 23(3):1-8.
- [6] Predin A., Biluš I., 2003, "Influence of additional inlet flow on the pre-rotation and performance of centrifugal impellers," Journal of Hydraulic Research, 41(2): 207-216.
- [7] Shao J., 2009, "The experiment research and numerical simulation on the unsteady internal flows of a small centrifugal model pump," Doctoral Dissertation of Tsinghua University, pp. 30-31.
- [8] Adrian R. J., 1991, "Particle-imaging techniques for experimental fluid mechanics," Annual Review of Fluid Mechanics, 23(1): 261-304.