A Study on Internal Flow in a New Type of Sewage Pump*

(1st Report, Comparison between Experiment and CFD Calculation)

Yasuyuki NISHI**, Nobuaki MATSUO*** and Junichiro FUKUTOMI***

**ShinMaywa Industries, Ltd.,

14 Takumidai, Ono-shi, Hyogo, 675-1322 Japan

E-mail: nishi.y@sa.shinmaywa.co.jp

***Department of Mechanical Engineering, The University of Tokushima,
2-1 Minamijosanjima, Tokushima, 770-8506 Japan

Abstract

Sewage pumps are designed with a wide flow channel by, for example, sacrificing some efficiency and reducing the number of blades, in order to prevent plugging with foreign bodies. Authors have been proposing a new type of sewage pump impeller designed to further improve pump efficiency and performance in passing foreign bodies. This sewage pump impeller has a structure in which the suction flow channel of closed type non-clog pump is wound in a helical spiral. The focus of this research was to investigate internal flow in this single blade sewage pump impeller. The results clearly indicated the following facts. As a result of having compared experimental results by LDV with calculated results by CFD, there is the difference about the radial component of absolute velocity, but agrees about pump performance and the tangential component comparatively well. A flow to circulate through in the impeller again is accepted by an experiment, and the sign is accepted by CFD in a specific impeller position.

Key words: Turbomachinery, Sewage Pump, Performance, Internal Flow, LDV, CFD

1. Introduction

Sewage pumps used in sewage treatment facilities transport fluids containing large amounts of various solids and fibrous foreign bodies, and thus it is desirable for pumps to have good performance in passing foreign bodies.

Various types of pump have been used as sewage pumps in recent years, including non-clog pumps⁽¹⁾⁽²⁾, screw pumps⁽¹⁾⁽³⁾, and vortex pumps⁽¹⁾; these pumps ensure a large passed particle size (the minimum particle size for the flow channel in the pump) in order to avoid clogging by foreign bodies. However, the behavior of foreign bodies actually flowing into a pump is extremely complex, and there is uncertainty about whether presumed foreign bodies actually pass through the pump. In addition, increasing the passed particle size causes the backflow at the blade inlet section and increased loss in the flow channel, thereby posing a problem in increasing the sewage pump efficiency. With this background, the authors proposed a new type of pump impeller for sewage use with the aim of improving the pump efficiency and performance in passing foreign bodies, showing that the performance characteristics are superior to those of conventional pumps and clarifying the behavior of radial thrust that poses design problems⁽⁴⁾. Furthermore, the authors investigated

*Received 9 July, 2009 (No. T1-07-0789)
Japanese Original : Trans. Jpn. Soc. Mech.
Eng., Vol.74, No.742, B (2008),
pp.1386-1394 (Received 6 Sep., 2007)
[DOI: 10.1299/jfst.4.648]

internal flow experimentally, clarifying its relationship with radial thrust⁽⁵⁾. However, the internal flow in this pump using a single blade is strongly unsteady, presenting a complex three-dimensional flow field that changes substantially depending on the impeller phase, with its particulars still remaining unknown in many respects.

On the other hand, in computational fluid dynamics (CFD) that has made remarkable progress in recent years, multi-blade impellers are commonly analyzed only on a single flow between blades. In the case of a single blade, however, according to reports⁽²⁾⁽³⁾, the flow field changes substantially depending on the impeller phase, causing strong mutual interference with the volute casing. For this reason, it is considered necessary, for this pump also, to perform unsteady analysis that takes into consideration the mutual interference between the impeller and the volute casing.

Accordingly, this study investigated the impeller outlet flow using a laser doppler velocimeter (LDV) in order to clarify the internal flow in the new sewage pump. Using general-purpose CFD codes, in addition, the authors performed unsteady flow analysis taking into consideration mutual interference between the impeller and the volute casing and surveyed the internal flow comparatively from the standpoints of both experiments and numerical analysis.

2. Nomenclature

- b : blade width [m]
- C_p : pressure coefficient $= (P - P_s) / (\rho u_2^2 / 2)$
- g : gravitational acceleration [m/s²]
- P : static pressure [Pa]
- P_t : total pressure [Pa]
- u : circumferential velocity [m/s]
- v : absolute velocity [m/s]
- η : pump efficiency $= \rho g Q H / L$
- λ : shaft power coefficient $= L / (\rho \pi D_2 b_2 u_2^3 / 2)$
- ρ : fluid density [kg/m³]
- ϕ : flow rate coefficient $= Q / (\pi D_2 b_2 u_2)$
- ϕ : head coefficient $= H / (u_2^2 / 2g)$
- Subscripts
- $1, 2$: blade inlet; blade outlet
- d, s : discharge side; suction side
- r, u : radial component; circumferential component
- x, y : x-direction component; y-direction component

3. Experimental apparatus and method

3.1 New type impeller

The new pump impeller for sewage use proposed in the previous report⁽⁴⁾ is outlined below. Inside the non-clog type impeller that increases the pressure of the working fluid by means of centrifugal action, a pressure gradient is formed to balance the centrifugal force. As a result, even if the passed particle size is increased, foreign bodies flowing into the impeller may be drawn into the region of low pressure near the center of rotation or into the region of flow separation and causes obstruction in some cases. To address this problem, the authors noted the structure of non-clog type impellers with which it is easy to secure the passed particle size and the structure of screw-type impellers that boosts the pressure of the working fluid through the volume and centrifugal actions, establishing the following basic

design conditions for the new impeller.

- (1) Ensure a passed particle size the same as the pump diameter.
- (2) Use a closed type impeller to prevent engagement of foreign bodies in the blade tip gap.
- (3) Shape the suction flow channel into a helical spiral to reduce obstruction by foreign bodies at the low pressure part in the impeller.
- (4) Continuously connect the backward inclined single blade to boost pressure of fluid which has passed through the spiral suction flow channel.

The new impeller designed on the basis of the above conditions attempts to improve the pump efficiency and the performance in passing foreign bodies by smoothening the turning of flow from the axial to the radial direction and at the same time, allowing foreign bodies to circle and flow in avoiding the center of rotation.

Fig. 1 shows a diagrammatic sketch of the new impeller. It has an outer diameter D_2 of 195 mm, a blade outlet angle β_2 of 15.1° (pressure side), a winding angle from the blade inlet end to the blade outlet end of 314° , and a winding angle from the suction inlet to the blade outlet of 591° . And it has a suction inlet diameter of 114 mm, a blade outlet width b_2 of 80 mm, and a discharge outlet diameter of 80 mm, ensuring a passed particle size of 80 mm for the passage from the suction inlet to the discharge outlet.

Fig. 1 New type impeller

3.2 Experimental apparatus and method

Fig. 2 shows a diagrammatic sketch of the experimental apparatus. The pressure on the suction side P_s and the pressure on the discharge side P_d were measured with a strain-gauge-type compact pressure transducer mounted to the static pressure holes which are in the suction and discharge pipes to determine the total pump head. The flow rate Q was varied with a gate valve mounted to the discharge side and measured with an electromagnetic flow meter. The rotation speed n was measured with an electromagnetic pickup and the driving torque T with a torsion-bar-type torque detector to determine the shaft power L . The experiment was carried out at a rotation speed n of 1740 min^{-1} .

Fig. 3 shows the definition of the coordinate system and the measurement points of velocity and pressure. The direction horizontal to the volute casing is defined as the X-axis and the direction perpendicular to the casing as the Y-axis, with the impeller phase angle θ_0 taken counterclockwise from the X-axis and set at 0° at the time when the blade outlet end passes the X-axis (the position shown in Fig. 3).

To measure velocity, a back-scattering one-dimensional laser doppler velocimeter with the beam source made up of 200 mW argon ion laser. This measurement method enables velocity data to be acquired only when microparticles pass a measurement point fixed in the space; this make it necessary to determine the relative position between the measurement point and the rotating blade in order to obtain the velocity distribution at the impeller outlet. Each rotation of the impeller causes a trigger signal to be generated, with the time of that moment being set at 0 each time. If a microparticle passes a measurement point while the

impeller makes a single rotation, the time elapsed from the generation of the trigger signal is obtained. The elapsed time and the rotational period of the impeller allowed the relative position between the blade and the measurement point to be determined.

Seven measurement points are provided, at intervals of 45° (not at this angle at some intervals) in the counterclockwise direction with the X-axis taken as $\theta = 0^\circ$, 10 mm radially apart from the blade outlet end. For a single measurement point, 10,000 measurements were made of v_u and v_r , respectively, being averaged with respect to the impeller phase angle θ_0 .

Pressure fluctuation was measured in the same way as velocity was measured, with a total of nine points set at the same measurement position except some. Pressure fluctuation during a single rotation of the impeller was measured in synchronism with the rotation of the impeller by generating a trigger signal when the blade outlet end is at $\theta_0 = 0^\circ$. The measurement accuracy was 0.8% or smaller for flow rate and about 3 to 4% for pressure and velocity.

Fig. 2 Experimental apparatus

Fig. 3 Measurement points

Fig. 4 Computational domain and grids

4. Method and conditions of analysis

4.1 Method of analysis

In numerical analysis, we used a general-purpose analysis code FLUENT 6.2 to carry out three-dimensional incompressible unsteady analysis. The basic equations used in this computational code are the mass conservation equation and the momentum equation that are solved by using the finite volume method. The standard wall function was used for

regions near wall surfaces and the standard $k-\epsilon$ model was adopted as the turbulence model.

4.2 Conditions of analysis

To take into consideration mutual interference between the impeller and the volute casing, this analysis uses an analysis model in which the volute casing is placed downstream of the impeller in order to perform computation. As shown in Fig. 4, the computational grid consists of a suction pipe, an impeller, volute casing, and a discharge pipe, with a total number of grids reaching about a million. To give boundary conditions, the inlet of the suction pipe is assumed to be at the inlet of the computational domain and the outlet of the discharge pipe at the outlet of the domain, with the mass flow rate in the absolute coordinate system given as the inlet boundary condition and a pressure of 0 Pa as the outlet boundary condition. Since the fluid domain for the impeller is in a rotational coordinate system, rotational angular velocity in the opposite direction to the experiment is given; the blade surface, and the wall surfaces of the rear shroud and front shroud are treated as fixed walls in a relative coordinate system; and the suction pipe, the volute casing, and the discharge pipe are set as fixed walls in the absolute coordinate system. For the connection between the outlet of the suction pipe and the suction port of the impeller and that between the outlet of the impeller and the inlet of the volute casing, the grid interfaces based on the sliding mesh method⁽⁶⁾ were used. This makes it possible to perform computation on an unsteady flow field.

5. Results of experiment and analysis and discussion

5.1 Comparison of pump performance characteristics

Fig. 5 shows the comparison in performance characteristics between the experimental values and the calculated ones obtained from flow analysis. The figure also shows the experimental values obtained by correcting leakage flow rate q , obtained by the method described in reference⁽⁴⁾, so that it may represent the case without leakage loss. The calculated value for the total pump head was determined from the difference between the total pressure averaged over the outlet boundary cross section and the total pressure averaged over the inlet boundary cross section. The calculated value for the shaft power was determined by integrating the pressure exerted on the blade surface and the wall shear stress to determine torque and adding to it the measured value of the torque in the case without water as mechanical friction loss. Aiming at the elucidation of internal flow, this study does not model the gap between the rear shroud and front shroud of the impeller and the volute casing and hence does not consider disk friction loss and leakage loss. The comparison of the head coefficient ϕ between the experimental and the calculated value shows that calculated ϕ values are slightly larger than experimental counterparts except in the domain of excess flow rate but they agree roughly with each other in the entire flow rate domain. The major reason why a calculated ϕ value is slightly larger than its experimental counterpart is the omission of consideration of leakage loss, and it is observed that an experimental ϕ value corrected for the absence of leakage loss comes closer to the calculated counterpart. Because of this effect, the calculated value for shaft power coefficient λ is slightly larger than its experimental counterpart in the partial flow rate domain in spite of the absence of consideration of disk friction loss. This is seen also from the fact that, although experimental ϕ values agree roughly with their calculated counterparts in the excess flow rate domain in which the effect of leakage loss is small, calculated values for shaft power coefficient λ are lower by the amount resulting from the absence of consideration of disk friction loss. The above discussion allows us to consider the result of analysis to be valid on the basis of generally good agreement between the calculated and the experimental pump efficiency η , though the former exhibits slightly larger values than the latter.

5.2 Time-averaged flow

5.2.1 Comparison of circumferential pressure distributions

Fig. 6 compares, for individual flow rates, the circumferential distribution of experimental values for time-averaged pressure just behind the blade outlet with that of the calculated values based on flow analysis. At the best efficiency point flow rate at which $\phi = 0.025$, both the experimental and the calculated value for pressure coefficient C_p are roughly constant in the circumferential direction. For a partial flow rate for which $\phi = 0.010$, however, both the experimental and the calculated value for C_p increase toward the volute winding end, while for an excess flow rate for which $\phi = 0.040$, C_p decreases toward the winding end. A comparison of both values shows that calculated ϕ values of 0.010 and 0.025 are slightly higher than their experimental counterparts. This is due to the absence of consideration of leakage loss; this corresponds to the fact that the calculated head coefficient ϕ shown in Fig. 5 is slightly higher than its experimental counterpart. With the presence or absence of leakage loss taken into consideration, experimental C_p values and calculated C_p values are in relatively good agreement.

Fig. 5 Performance curves

Fig. 6 C_p distributions in time average

5.2.2 Comparison of circumferential velocity distributions

Figs. 7 and 8 compare, for individual flow rates, the circumferential distribution of experimental time-averaged v_u and v_r values at the center of the impeller outlet width with that of the calculated values based on flow analysis, respectively. Fig. 7 shows that both experimental and calculated v_u values slightly increase from the volute winding start toward the end at the best efficiency point flow rate at which $\phi = 0.025$ but they remain roughly constant in the circumferential direction. At a partial flow rate for which $\phi = 0.010$, v_u decreases toward the volute winding end while increasing toward the winding end at an excess flow rate for which $\phi = 0.040$; this corresponds to the circumferential pressure distribution shown in Fig. 6. Comparing v_u values in both cases shows a generally good agreement except a slight difference seen near the volute winding start at which $\phi = 0.040$.

Fig. 8 shows that for $\phi = 0.040$, experimental v_r values become negative near $\theta = 130^\circ - 190^\circ$ and that they are relatively high near $\theta = 40^\circ - 110^\circ$ and $220^\circ - 360^\circ$. In other words, when phenomena are seen on a time-averaged basis, a circulating flow is formed in which the fluid in the volute flows into the impeller when θ reaches the range near $130^\circ - 190^\circ$, flows across the impeller and flows out when θ reaches the range near $40^\circ - 110^\circ$ and a pass-through flow is formed which flows out toward the discharge outlet when θ reaches the range near $220^\circ - 360^\circ$. Figs. 9 (a) and (b) show the results of analysis of absolute velocity vectors at an impeller phase angle θ_0 of 45° in two cases for which $\phi = 0.025$ and $\phi = 0.040$, respectively. The cross section is at the center of the width, $b/b_2 = 0.5$. As described below, v_r exhibits large changes as θ_0 changes; from Fig. 9 (b), however, we can see a flow

that, when θ is near 150° , flows into the impeller from the volute and another that flows out from the impeller to the neighborhood of the volute winding start and to the discharge outlet. When the flow rate increases beyond that at the best efficiency point, part of volute flow goes into the main impeller provided with wide flow channels; for this reason, it is considered that this trend becomes remarkable during excess flow rate.

Comparison between experimental and calculated values of v_r shows that both exhibit roughly similar values near the volute winding start of $\theta = 0 - 135^\circ$ for $\phi = 0.025$; however, the degree of agreement becomes poor from there to the vicinity of the winding end. For $\phi = 0.040$, the difference in v_r between experimental and calculated values is large near the volute winding start of $\theta = 45^\circ - 90^\circ$, and roughly similar values are exhibited from there to the vicinity of the winding end. However, in v_u values for $\phi = 0.040$ shown in Fig. 7, difference between experimental and calculated values is observed when θ is near the range of $45^\circ - 90^\circ$. The stagnation point is located near the tongue section, at which a flow that comes out from the impeller and directly runs toward the discharge outlet under the condition of $\phi = 0.025$, as shown in Fig. 9 (a), and another that recirculates inside the volute bifurcate. In contrast to this, under the condition of $\phi = 0.040$ as shown in Fig. 9 (b), the stagnation point is located near the volute winding start considerably away from the tongue section, and a flow that runs from there in the reverse direction toward discharge outlet and another that flows from the impeller interfere with each other, forming a very complex flow field. Accordingly, it is estimated that this analysis was not capable of adequately capturing the flow near the volute winding start at $\theta = 45^\circ - 90^\circ$ in the case of $\phi = 0.040$, an excess flow rate state, with difference in v_u and v_r occurring between the experimental and the calculated value. In addition, for $\phi = 0.010$, experimental v_r values are nearly uniform in the circumferential direction, but their values are the largest in spite of

Fig. 7 v_u distributions in time average ($b/b_2=0.5$)

Fig. 8 v_r distributions in time average ($b/b_2=0.5$)

Fig. 9 Absolute velocity vectors distributions ($\theta=45^\circ$)

small flow rate. It is known that if a three-dimensional backflow occurs near the wall surface, v_r becomes very large at the center of the width⁽⁷⁾, and the experiment suggests that the fluid flows out from the center of the width locally. A comparison of v_r with v_u shown in Fig. 7 reveals that v_r is much smaller than v_u and v_u is predominant in the flow field of this pump.

5.2.3 Comparison of width direction velocity distributions

Figs. 10 and 11 compare, for individual flow rates, the width direction distribution of experimental time-averaged v_u and v_r values at $\theta = 270^\circ$ with that of the calculated values based on flow analysis, respectively. The measurement on the hub side was not performed due to problems with the experiment apparatus. Fig. 10 shows that both experimental and calculated v_u values increase as the flow rate increases. Under the condition of $\phi = 0.025$, calculated v_u values are slightly smaller at the center of the width and larger on the shroud and the hub side. Although experimental v_u values on the hub side are unknown, the experimental and the calculated values elsewhere exhibit a similar trend. However, at partial flow rate and excess flow rate, calculated v_u values decrease slightly from the shroud side to the hub side. This trend is observed with experimental values obtained in the range in which measurement has been performed.

Fig. 11 shows that for $\phi = 0.040$, both experimental and calculated v_r values are the smallest at the center of the width and increase toward the shroud and the hub side. A comparison between experimental and calculated v_r values shows that they are roughly close to each other. However, as the flow rate decreases, experimental v_r values become larger at the center of the width, showing that the fluid flows outward from the center of the width locally as described above under the condition of $\phi = 0.010$. It is known that the standard $k-\epsilon$ model used in this study suffers a decline in calculation accuracy with strong swirling flow. In partial flow rate, in particular, difference between experimental and calculated v_r values in the circumferential and width directions are observed; the estimated reason for this is that strong swirling flow causes the standard $k-\epsilon$ model to suffer a decline in calculation accuracy.

From the above discussion, it is clear that, although difference between the experimental and the calculated v_r value is observed, v_u is predominant in the flow field of this pump with v_r being considerably smaller than v_u . For v_u values that determine the pump head, both experimental and calculated values are in considerably good agreement in the circumferential distribution as well as in the width direction distribution at all flow rates. In addition, an overall approximate agreement is observed in terms of the circumferential pressure distribution and the pump head curve. Accordingly, it is concluded that this analysis roughly reproduces the time-averaged flow field of this pump.

Fig. 10 v_u distributions in time average ($\theta = 270^\circ$)

Fig. 11 v_r distributions in time average ($\theta = 270^\circ$)

5.3 Flow during a single rotation of the impeller

5.3.1 Comparison in pressure fluctuation

Fig. 12 compares, for individual flow rates, experimental values for pressure fluctuation during a single rotation of the impeller with their counterpart calculated values based on flow analysis. The measurement position is at $\theta = 0^\circ$. Fig. 12 shows that for all flow rates, C_p lowers before and after the passage of the blade outlet end; this trend is captured by experimental and calculated values alike.

A comparison between the experimental and the calculated value shows that they exhibit rough agreement for all flow rates.

Fig. 12 C_p distributions

5.3.2 Comparison of velocity fluctuations at various circumferential positions

Figs. 13 (a) and (b) and Figs. 14 (a) and (b) each show, for individual flow rates, the comparison of experimental v_u and v_r values at various circumferential positions at the center of the width during a single rotation of the impeller with the counterpart calculated values based on flow analysis. Figs. 13 (a) and (b) show that the minimum v_u value occurs just before the passage of the blade outlet end for all flow rates. This trend is captured by experimental and calculated values alike. A comparison between experimental and calculated v_u values show that both v_u values are in considerably good agreement.

It is seen from Fig. 13 (a) that under the condition of $\phi = 0.025$, v_u is nearly constant at $\theta = 135^\circ$ and 270° regardless of θ_0 . At the measurement point of $\theta = 0^\circ$, v_u lowers to the smallest value near $\theta_0 = 350^\circ$. In addition, Fig. 13 (b) shows that the above trend is more remarkable under the condition of $\phi = 0.040$ with the v_u value approaching zero; at $\theta_0 = 350^\circ$, the v_r value shown in Fig. 14 (b) has increased, which suggests that the flow is pulled in toward the discharge outlet. And the v_u distributions at each measurement point during a single rotation of the impeller exhibit few intersections, which indicate that the circumferential position θ of v_u in the volute exerts a greater effect. This is understood also from the circumferential change in the v_u value for $\phi = 0.040$ shown in Fig. 7.

Figs. 14 (a) and (b) show that in both experimental and calculated values, v_r decreases sharply just before the passage of the blade outlet end at any flow rate. There is a domain in which v_r increases a little immediately after the passage of the blade outlet end but, unlike v_u shown in Fig. 13, starts decreasing gradually. For this reason, the minimum v_r value during a single rotation of the impeller exists not in the wake of the blade outlet end but in the domain described above. After this, v_r increases again toward the blade winding end. The largest increment of this v_r occurs at the measurement point of $\theta = 0^\circ$ under the condition of $\phi = 0.040$. At this measurement point, v_r exhibits the maximum near the range of $\theta_0 = 310^\circ - 350^\circ$ at all flow rates, becoming negative at the measurement point of $\theta = 135^\circ$. In other words, while the blade end discharges a large quantity of fluid in the discharge direction near the volute winding end, it draws in the fluid on the other side conversely.

When this phenomenon is observed on a time-averaging basis, it would take the form of a through-flow shown in Fig. 8.

A comparison between experimental and calculated v_r values shows that, under the condition of $\phi = 0.025$, the decrease in v_r just before the passage of the blade outlet end was captured, showing rough qualitative agreement. It is further known that both are in relatively good agreement under the condition of excess flow rate $\phi = 0.040$. It has been additionally observed that a comparison between experimental and calculated values for v_u and v_r under the condition of $\phi = 0.010$ shows that although the degree of agreement in v_r values is poorer than under the condition of $\phi = 0.025$, v_u values are in relatively good agreement.

(a) $\phi = 0.025$

(a) $\phi = 0.025$

(b) $\phi = 0.040$

(b) $\phi = 0.040$

Fig. 13 v_u distributions ($b/b_2=0.5$)

Fig. 14 v_r distributions ($b/b_2=0.5$)

5.3.3 Comparison of velocity fluctuations at various positions in width direction

Figs. 15 (a) and (b) and Figs. 16 (a) and (b) each show, for individual flow rates, the comparison of experimental v_u and v_r values at several positions in width direction at $\theta = 270^\circ$ during a single rotation of the impeller with the counterpart calculated values based on flow analysis. Figs. 15 (a) and (b) show that, in both experimental and calculated values, the minimum v_u value occurs, independently of the width direction position, just before the passage of the blade outlet end for all flow rates. A comparison of width direction positions shows that the minimum v_u during a single rotation of the impeller occurs at the measurement point of the center of the width at $b/b_2 = 0.5$. Following this, the minimum v_u value occurs at $b/b_2 = 0.125$ on the shroud side and at $b/b_2 = 0.875$ on the hub side, in ascending order. There are domains in which the v_u value decreases to the minimum and then increases before it decreases again. The greater the flow rate becomes, the more remarkable this trend is. After this, v_u increases again toward the blade winding end, and

(a) $\phi = 0.025$

(a) $\phi = 0.025$

(b) $\phi = 0.040$

Fig. 15 v_u distributions ($\theta = 270^\circ$)

(b) $\phi = 0.040$

Fig. 16 v_r distributions ($\theta = 270^\circ$)

Fig. 17 Total pressure distribution at the impeller exit ($\phi = 0.025$, $r/r_2 = 1.1$, $\theta_0 = 0^\circ$)

then it decreases sharply just before the passage of the blade outlet end. A scrutiny into the case of $\phi = 0.025$ reveals that, in this domain, the v_u value is greater on both shroud side at $b/b_2 = 0.125$ and hub side at $b/b_2 = 0.875$ than at the center of the width at $b/b_2 = 0.5$. As described later, this is considered due to the fact that the pressure surface of this impeller is formed by a semicircular flow channel. A comparison between experimental and calculated v_u values shows that the trend described above is captured by both experiments and analysis and that the experimental and the calculated values are in agreement qualitatively.

Figs. 16 (a) and (b) show next that, in both experimental and calculated values, v_r decreases sharply, independently of the width direction position, just before the passage of

the blade outlet end for all flow rates. After this, like v_u , v_r exhibits an increase followed by a decrease, and again an increase toward the blade winding end. A comparison of width direction positions shows that it is at the center of width at $b/b_2 = 0.5$ that the sharp decrease in v_r just before the passage of the blade outlet end occurs least quickly. In addition, it is at the measurement point at the center of the width at $b/b_2 = 0.5$ that the minimum v_r value during a single rotation of the impeller exhibits the lowest value. However, as the blade winding end is approached, the v_r value at $b/b_2 = 0.5$ increases sharply, exhibiting the largest value in the width direction position at θ_0 where the v_r value on the shroud side and that on the hub side start decreasing. The possible reasons why these occur are that, with the pressure surface of this impeller formed by a semicircular flow channel delineated by the locus of a sphere, the blade width is small in the direction toward the winding end of the blade pressure surface and the flow spills from the center of the width locally. A comparison between experimental and calculated v_r values shows that the trend described above is captured by both experiments and analysis and that the experimental and the calculated values are in agreement qualitatively.

The above discussion allows us to estimate that this analysis roughly reproduces the flow field during a single rotation of the impeller of this pump in the range from the best efficiency point to the excess flow rate.

Fig. 17 shows the total pressure distribution based on flow analysis at the impeller outlet at $r/r_2 = 1.1$ under the condition of $\phi = 0.025$. The impeller phase angle θ_0 is equal to 0° and the total pressures near the wall surface on the shroud and hub side are not given because of lack of data. An observation of the section between the hub and the shroud shows that the total pressure is higher on the shroud and the hub side than at the center of the width for the same circumferential position θ . This trend is seen also from the calculated v_u values shown in Fig. 15 (a) and (b); it is seen also from the comparison of width direction positions for individual flow rates. This characteristic also represents the feature of this impeller well; with the pressure surface formed by the semicircular flow channel, it is estimated that the larger blade diameter on the shroud and the hub side contributes to better work efficiency. An observation of phenomena occurring between blades shows that while the total pressure rises over the transition from the negative pressure surface side to the pressure surface side, the total pressure abruptly falls over the transition from a point near $\theta = 75^\circ$ toward the pressure surface side. This will be discussed in detail in the second report.

6. Conclusion

We have studied internal flow in a new type of sewage pump from the viewpoints of both experiments and numerical analysis, acquiring the following findings:

(1) Unsteady flow analysis taking the mutual interference between the impeller and the volute casing into consideration can predict the pump performance over most of the flow rate range.

(2) A comparison between experimental and calculated values, in a time-averaged flow field and a flow field during a single rotation of the impeller, reveals difference in the radial component of absolute velocity; but it also shows that the circumferential component of absolute velocity dominating the head rise of this pump and the resulting static pressure distribution are in relatively good agreement.

(3) While the blade end discharges a large quantity of fluid in the discharge direction near the volute winding end, it draws in the fluid on the other side conversely. An experiment based on a time-averaging viewpoint ascertains that the fluid in the volute appears in forms of a circulating flow that enters the impeller, flows across it and goes out from a point near the winding start and a through-flow that goes out from a point near the winding end and flows toward the discharge outlet, with an indication of this event

observed at specific blade positions in the analysis.

(4) Since the pressure surface of this impeller is formed by a semicircular flow channel, the total pressure is higher on the shroud and the hub side than at the center of the width.

References

- (1) Japan institute of Wastewater Engineering Technology, *Wastewater Manhole Pump Technology Manual-June 1997*, (1997), p. 18 (in Japanese).
- (2) Aoki, M., *Transactions of the Japan Society of Mechanical Engineers, Series B*, Vol.50, No.451 (1984), p. 661 (in Japanese).
- (3) Han, H., Tanaka, K. and Kobayashi, T., *Turbomachinery*, Vol.28, No.3 (2000), p. 183 (in Japanese).
- (4) Nishi, Y., Makita, K. and Fukutomi, J., *Transactions of the Japan Society of Mechanical Engineers, Series B*, Vol.72, No.720 (2006), p. 1984 (in Japanese).
- (5) Nishi, Y., Makita, K. and Fukutomi, J., *Turbomachinery*, Vol.34, No.12 (2006), p. 729 (in Japanese).
- (6) *Fluent User's Manual*, (2006), pp. 9-30 (in Japanese).
- (7) Kurokawa, J. and Hode, S., *Transactions of the Japan Society of Mechanical Engineers, Series B*, Vol.50, No.459 (1984), p. 2777 (in Japanese).