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

(2nd Report, Influence of Impeller Blade Loading and Volute Casing)

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 by LDV measurement and CFD. As a result of having examined influence of impeller blade loading and volute casing, it clearly indicated the following facts. As well as influence of impeller blade middle loading and influence of the secondary flow, by increase of slip by influence of volute casing, static pressure and total pressure suddenly decrease together towards the end from impeller blade middle neighborhood.

Key words: Turbomachinery, Sewage Pump, Internal Flow, Impeller Blade Loading LDV, CFD

1. Introduction

Sewage pumps carry fluids containing large amounts of various solid and fibrous foreign bodies, and for this reason, their performance in passing foreign bodies is regarded as important. Various types of pump have been used as sewage pumps, 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 passes through the pump. In addition, increasing the passed particle size results in 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 internal flow experimentally, clarifying its relationship with radial thrust⁽⁵⁾. However, the

*Received 24 July, 2009 (No. T2-07-0404)
Japanese Original : Trans. Jpn. Soc. Mech.
Eng., Vol.74, No.743, B (2008),
pp.1496-1504 (Received 27 Apr., 2007)
[DOI: 10.1299/jfst.4.661]

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 mutual interference between the impeller and the volute casing.

As described in the first report⁽⁶⁾, the authors investigated impeller outlet flow by means of a laser doppler velocimeter (LDV), and using the general-purpose CFD codes, they performed unsteady flow analysis that takes into consideration mutual interference between the impeller and the volute casing. And by comparing the experimental result with the result of analysis, they demonstrated that this analysis roughly reproduces internal flow in this pump. In this report, the authors discuss internal flow more detailedly. At the same time, assuming that the flow in the impeller under the design flow rate is in a free impeller state without a volute, they analyze steady flow using an analysis model with a parallel-walled diffuser placed downstream of the impeller to examine the effect of impeller blade loading and the volute casing.

2. Nomenclature

b	: blade width [m]
C_p	: pressure coefficient $= (P - P_s) / (\rho u_2^2 / 2)$
g	: gravitational acceleration [m/s ²]
P	: static pressure [Pa]
P_t	: total pressure [Pa]
u	: circumferential velocity [m/s]
v	: absolute velocity [m/s]
w	: relative velocity [m/s]
β	: relative flow angle [deg]
η	: pump efficiency $= \rho g Q H / L$
θ_a	: angle measured anticlockwise from the blade outlet end [deg]
λ	: shaft power coefficient $= L / (\rho \pi D_2 b_2 u_2^3 / 2)$
ρ	: fluid density [kg/m ³]
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 basic design conditions for the new pump impeller for sewage use proposed in the previous report⁽⁴⁾ are outlined below:

- (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 while getting around the center of rotation.

Fig. 1 shows a diagrammatic sketch of the new impeller. The new impeller 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.

3.2 Experimental apparatus and method

Fig. 2 shows a diagrammatic sketch of the experimental apparatus. The total pump head H was determined from the difference between the total pressure in front of the pump and that after the pump, while the flow rate Q was 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 was used. The relative position between the blade and the measurement point was obtained by means of the time of one cycle determined on the basis of the number of rotations and the time elapsed from the moment of the generation of a trigger signal generated for every rotation of the impeller. Seven measurement points are set, 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 reference 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 $\theta_0 = 0^\circ$.

4. Method and conditions of analysis

4.1 Method of analysis

The general-purpose analysis code FLUENT6.2 was used in numerical analysis. The basic equations used in this computational code are the mass conservation equation and the conservation of momentum equation, which are solved by using the finite volume method. The standard wall function was used for regions near wall surfaces and the standard k- ϵ model was adopted as the turbulence model.

Fig. 1 New type impeller

Fig. 2 Measurement points

Fig. 3 Experimental apparatus

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 unsteady flow analysis. As shown in Fig. 4, the computational grid consists of a suction pipe, an impeller, a 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 region of the impeller is expressed in the rotational coordinate system, rotational angular velocity in the opposite direction to that adopted in the experiment is given; the blade surface, and the wall surfaces of the rear shroud and front shroud are defined in the relative coordinate system; and the fixed wall, the suction pipe, the volute casing, and the discharge pipe are defined on the fixed wall in the absolute coordinate system. The outlet of the suction pipe, the suction port and outlet of the impeller, and the inlet of the volute casing are connected by grid interfaces; this makes it possible to perform computation on a unsteady flow field by means of the sliding mesh method⁽⁷⁾.

To investigate the internal flow in the impeller in isolation from others, the authors assumed that the flow inside the impeller under the design flow rate is in a state of a free impeller without a volute, using an analysis model with a parallel-walled diffuser three times as large as the outside diameter of the impeller placed downstream of it to try to

analyze steady flow. As shown in Fig. 5, the computational grid consists of a suction pipe, an impeller, and a parallel-walled diffuser, with a total number of grids reaching about 1.06 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 parallel-walled diffuser 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. It is possible that the fluid flowing from the impeller loses momentum through the friction with the wall surface in a long parallel-walled diffuser stretch, and the separated flow runs in the reverse direction and affects the flow at the impeller outlet. Accordingly, we assumed that the wall surface of the parallel-walled diffuser is slippery. Since the fluid region of the impeller is expressed in the 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 defined in the relative coordinate system; and the fixed wall, the suction pipe, and the parallel-walled diffuser are defined on the fixed wall in the absolute coordinate system.

Fig. 4 Computational domain and grids

Fig. 5 Computational domain and grids

5. Results of experiment and analysis and discussion

5.1 Comparison of pump performance characteristics

Fig. 6 compares the experimental values, the unsteady and the steady analytical values of pump performance characteristics. The total pump head in a unsteady analytical value was determined from the difference between the mean cross section total pressure at the outlet boundary and that at the inlet boundary, while the total pump head in a steady analytical value was determined from the difference between the mean flow rate total pressure at the position $r/r_2 = 1.1$ and the mean cross section total pressure at the inlet boundary. The shaft power in a unsteady analytical value and that in a steady analytical value were 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 empty torque as mechanical friction loss. Aiming at the elucidation of internal flow, this study does not intend to model the gap between the rear shroud and front shroud of the impeller and the volute casing.

A comparison between the experimental values and the unsteady analytical ones shows that head coefficient ϕ values obtained in unsteady analysis 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. A slightly higher ϕ value obtained in unsteady analysis is due to the absence of the consideration of leakage loss as described in the first report. A comparison between the experimental values and the steady analysis values shows that the ϕ values obtained in the steady analysis are slightly smaller than the

experimental values with the absence of the volute loss and leakage loss taken into consideration. This is considered due to an increase in slip, as shown in Fig. 17 later, resulting from a region in which the outlet flow runs into the impeller reversely.

As a result, the shaft power coefficient λ obtained in the steady analysis is slightly lower than the experimental value in spite of the omission of consideration of the disk friction loss. However, with the pump efficiency η obtained in steady analysis being larger than the experimental value, though slightly, the result of the steady analysis is considered valid as a whole. The above findings suggest that this steady flow analysis assuming a state of a free impeller enables the pump performance under the design flow rate to be estimated in a simple way and the flow field of an impeller in isolation from others to be roughly reproduced.

Fig. 6 Performance curves

5.2 Comparison of time-averaged pressure distributions

In the first report, the authors showed that the experimental values and the unsteady analytical values agree with each other roughly concerning the time-averaged flow and the flow during a single rotation of an impeller, demonstrating that this analysis reproduces the flow field of this pump roughly. In this report, we will make comparative studies on the pressure field including steady analytical values to further investigate the impeller blade loading. Fig. 7 compares the experimental values, the unsteady and the steady analytical values of the time-averaged pressure immediately behind the blade outlet under the condition of the best efficiency point flow rate of $\phi = 0.025$. The steady analytical value is the circumferential average value of the static pressure in the vicinity of the shroud side wall. Both experimental C_p values and unsteady analytical C_p values are nearly constant in the circumferential direction. This suggests that the assumption of a state of a free impeller holds at the design flow rate in this impeller with a single blade also.

A comparison between the experimental value and the unsteady analytical ones shows that C_p values obtained by the unsteady analysis exhibits slightly higher values. This is due to the absence of consideration of leakage loss in the analysis, and if it is taken into consideration, the experimental values and the analytical ones agree with each other roughly. Although experimental values lie between steady analytical values and unsteady analytical ones, they are slightly close to steady analytical values.

5.3 Comparison of pressure fluctuation during a single rotation of the impeller

Fig. 8 compares, for the condition of $\phi = 0.025$, experimental values of pressure change during a single rotation of the impeller with their counterpart calculated values based on the unsteady analysis. Fig. 8 shows that, in the experimental values and their counterpart unsteady analytical values alike, the minimum value of the pressure coefficient C_p appears near the passage of the blade outlet end for all measurement points. This results, as described later, from a decrease in the static pressure that occurs as the winding end on

the blade pressure surface is approached and from the effect of the negative pressure surface; this trend is captured by both the experimental values and the unsteady analytical ones. A comparison of both shows that the unsteady analytical values are slightly greater than the experimental ones as a whole. This is due to the absence of the consideration of leakage loss in the unsteady analytical values as described above, but quantitatively, the experimental values and the unsteady analytical ones are in good agreement. The above consideration suggests that the time-averaged pressure field and the pressure field during a single rotation of the impeller are reproduced by this analysis satisfactorily.

Fig. 7 C_p distributions in time average ($\phi=0.025$)

Fig. 8 C_p distributions ($\phi=0.025$)

5.4 Internal flow state

Figs. 9 (a) – (d), 10 (a) – (d), and 11 (a) – (d) show the results of the unsteady flow analysis on the absolute velocity vector, the static pressure distribution, and the total pressure distribution at different impeller phase angles θ_0 under the best efficiency point flow rate with $\phi=0.025$. The cross section is located at the center of the width with $b/b_2=0.5$. A scrutiny on Figs. 9 (a) – (d) from the viewpoint of the flow near the tongue section makes it clear that lines of flow bifurcate near the end of the tongue. Figs. 10 (a) – (d) show that the C_p value near the tongue end section is large and a stagnation point is formed; accordingly, it is clear that the volute flow of this pump is compatible with the orientation of the tongue end section. The distribution of static pressure in Fig. 10 (a) shows that the C_p value in the direction of $\theta=90^\circ$ is large under the condition of the impeller phase angle of $\theta_0=0^\circ$. It is easily understood from Figs. 10 (b) – (d) that this high pressure region rotates with the change in the impeller phase angle θ_0 . The absolute velocity vector in Fig. 9 (a) shows that the absolute velocity v on the blade surface on the straight line from this high pressure region to the rotational center is considerably large. In the flow field of this impeller, v_r is considerably smaller than v_u , which is hence predominant. Accordingly, this high pressure region is considered to be due to the impeller blade loading; we will examine this in detail in the following.

Fig. 12 compares the simplified calculated values and the unsteady analytical values of the impeller blade loading distribution at the center of the width with $b/b_2=0.5$ under the condition of $\phi=0.025$. The simplified calculated values were obtained by means of the method reported in the previous report⁽⁴⁾. With the slip not taken into consideration here, the rate of change of the angular momentum and the pressure difference between the blade surface and the back do not indicate zero at $\theta_a=0^\circ$. Fig. 12 suggests that unsteady analytical values are affected by the volute; it is clear, however, that the rate of change in the simplified calculated angular momentum near $\theta_a=180^\circ$ is large and accordingly, the pressure difference between the surface and the back of the blade in the simplified calculated values and that in the unsteady analytical values are large. The location at which

the rate of change in this angular momentum is large corresponds roughly to the location in Fig. 9 (a) at which v_u is high. In other words, an increase in the circumferential component v_u of the absolute velocity originating in the impeller blade loading causes the pressure on the blade surface to increase, and from there, v_u that forms a free vortex distribution in the

Fig. 9 Absolute velocity vectors ($\phi=0.025$)

Fig. 10 Static pressure distributions ($\phi=0.025$)

Fig. 11 Total pressure distributions ($\phi=0.025$)

Fig. 12 Impeller blade loading distributions ($\phi=0.025$)

Fig. 13 Static pressure distributions ($\phi=0.025$)

Fig. 14 Total pressure distributions ($\phi=0.025$)

radially outward direction decreases; and in contrast, C_p rises by the amount of the increment of centrifugal force to attain a high pressure. For these reasons, v_u that forms a free vortex distribution in the radial outward direction from the impeller outer diameter increases, and this overlaps the domain with large C_p values, exhibiting a total pressure distribution shown in Fig. 11 (a).

In addition, as shown in Fig. 9 (a), v_u is very large in the vicinity of the winding start of the blade pressure surface, because the blade angle is large there like a radial blade. As a result, the total pressure on the same radius behind the blade outlet end becomes high. Outside there, however, the total pressure falls sharply, probably causing losses.

Depending on the impeller blade loading distribution, both static and total pressure increase toward the blade winding end in ordinary centrifugal impellers. In this impeller also, the steady flow analysis on the case in which a parallel-walled diffuser is installed downstream of the impeller as shown in Fig. 13 show that C_p increases as the blade winding end is approached. A similar trend is observed in the total pressure distribution shown in Fig. 14, though the increase in the total pressure toward the winding end is slower than the increase in the static pressure. As shown in Figs. 10 (a) and 11 (a), however, the analysis on a unsteady flow in which this impeller and the volute casing are combined shows that both static and total pressure decrease in the range from the vicinity of the middle of the blade to the blade winding end, which is contrary to the above observation. The sharp decrease in the total pressure toward the blade winding end is attributable to a decrease in both absolute velocity v and C_p as well as to the fact, as an important cause, that the impeller blade loading distribution of this impeller is of a medium load type. In addition, the sharp decrease in the absolute speed v near the blade winding end suggests that the sharp decrease is due to a sharp increase in the relative velocity w .

To help examine this more detailedly, Figs. 15 and 16 show, for the condition of $\phi = 0.025$, a comparison between the experimental values and their counterpart unsteady analytical ones for the relative velocity w and that for the relative flow angle β , during a single rotation of the impeller, respectively. Based on Fig. 15, a comparison of w between the experimental and the unsteady analytical shows that they agree with each other relatively well. A scrutiny of unsteady analytical values reveals that, at all measurement points, w increases sharply immediately before the passage of the blade outlet end and decreases slowly after the passage, indicating a velocity distribution close to that of non-viscous flow. Although w increases immediately before the passage of the blade outlet end, namely, near the blade winding end, the experimental and the unsteady analytical values for the relative flow angle β shown in Fig. 16, too, exhibit a sharp increase near the blade winding end. The cause of this phenomenon is considered due to the fact that, as a result of secondary flow generated by Coriolis force, a jet region exists on the blade pressure surface side and causes more flow to run along the pressure surface⁽⁸⁾. Under the influence of this secondary flow, the relative velocity increases while C_p decreases. However, an increase in C_p toward the blade winding end shown in the result of analysis of steady flow in the absence of the volute, shown in Fig. 13, suggests the presence of the effect of the volute casing.

Fig. 17 compares, for the condition of $\phi = 0.025$, experimental values, unsteady analytical ones, and steady analytical ones of the slip factor during a single rotation of the impeller, respectively. The slip factor k was determined on the basis of v_{u2} at the center of the width at $\theta = 0^\circ$. A comparison between experimental and unsteady analytical values shows that both capture a decrease in k near $\theta_0 = 330^\circ$, namely, just in front of the blade outlet end and that they agree with each other roughly. A comparison between unsteady and steady analytical values shows that the k value based on steady analysis decreases sharply near $\theta_0 = 330^\circ$ while that based on unsteady analysis exhibits a smaller decrease in k values. Accordingly, slip becomes larger in the case where a volute exists than in the case where it

does not exist, with the relative velocity w increasing consequently. In particular, the abrupt decrease in the static and the total pressure in the portion from the vicinity of the middle of the blade to the blade winding end becomes more remarkable as θ_0 decreases. In other words, it is suggested that, since the flow channel cross section formed by the impeller and the volute casing decreases as the volute winding start is approached, the relative velocity w is increased as the blade winding end is approached, with both static and total pressure falling sharply. Furthermore, the increase in this relative velocity w and the resulting abrupt change in the curvature at the blade pressure surface winding end as shown in Fig. 10 (a) causes C_p to decrease and local separation to occur. The occurrence of this local separation can be understood from the fact that, as shown in Fig. 11 (a), a region in which the total pressure decreases exists slightly behind the one in which C_p decreases. In contrast to what occurs on the blade pressure surface, the reverse pressure gradient is steep toward the outlet on the negative pressure surface side, but C_p increases smoothly. It is considered, therefore, that the accumulation of low energy fluid seldom occurs in spite of a decrease in the relative velocity w . In this way, C_p on the negative pressure surface side increases and comes to balance with C_p on the pressure surface side that has dropped sharply toward the blade winding end as described above. It is considered from the above consideration that a jet region exists on the pressure surface side and a wake region on the negative pressure surface side under the influence of the secondary flow due to Coriolis force, because the single blade configuration of this impeller allows the slip to have strong influence, in particular because the influence of the slip is relatively strong near the blade winding end due to the effect of the volute casing.

Fig. 15 w distributions ($\phi=0.025$, $b/b_2=0.5$)

Fig. 16 β distributions ($\phi=0.025$, $b/b_2=0.5$)

Fig. 17 Slip factor distributions ($\phi=0.025$, $b/b_2=0.5$)

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) The blade angle near the blade pressure surface winding start is as large as that of a radial blade; this results in a remarkably large circumferential component of the absolute velocity. As a result, the total pressure becomes higher on the same radius behind the blade outlet end, but it decreases rapidly outside that radius.

(2) The impeller blade loading causes the circumferential component of the absolute velocity to increase; this in turn causes the pressure on the blade surface to increase. The high pressure region that is raised from there radially outward by centrifugal force rotates with the impeller.

(3) Not only the effect of the load on the blade middle and that of the secondary flow but also an increase in slip under the influence of the volute casing cause both static and total pressure to decrease sharply from the vicinity of the blade middle toward the winding end.

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) Nishi, Y., Matsuo, N. and Fukutomi, J., *Transactions of the Japan Society of Mechanical Engineers, Series B*, Vol.74, No.742 (2008), p. 1386 (in Japanese).
- (7) *Fluent User's Manual*, (2006), pp. 9-30 (in Japanese).
- (8) Johnston, J. P. and Dean, R. C., *Transactions of the American Society of Mechanical Engineers, Journal of Engineering for Power*, Vol.49, No.62 (1966), p. 88.