Investigation of Internal Flow in Centrifugal Pump Diffuser using Laser Doppler Velocimetry (LDV) and Computational Fluid Dynamics

Daisuke Sugiyama¹, Asuma Ichinose¹, Tomoki Takeda¹, Kazuyoshi Miyagawa², Hideyo Negishi³, Atsuhiro Tsunoda¹

¹Department of Applied Mechanics, Graduate School of WASEDA University, Okubo3-4-1 Shinjuku-ku Tokyo Japan
²Departments of Applied Mechanics and Aerospace Engineering, WASEDA University
³Japan Aerospace Exploration Agency

Abstract. An unshrouded impeller is being developed for high head pumps to reduce costs and disk friction losses. On the other hand, research of internal flow in a diffuser did not clearly reveal flow structure. In this experiment, we measured the velocity distribution at the diffuser inlet and outlet plane using laser doppler velocimetry (LDV) to capture the flow from the unshrouded impeller. The relation between the total pressure and velocity distribution was evaluated. The circumferential velocity and meridian velocity were measured by short-focus LDV (Diode Laser, 74mW) about the circumferential and the height direction of the vane direction. Operating conditions in this steady measurement are at the design point flow rate. The result was compared with computational fluid dynamics (CFD) simulations carried out in steady conditions at the previously defined operation points. In this experiment, a phenomenon that the streamline moved toward the shroud side was confirmed. There was also a region where the static pressure increased on the shroud side at the diffuser inlet. This phenomenon was caused by the influence of the tip leakage flow of the unshrouded impeller downstream and the gap of the impeller’s main plate. Furthermore, two high velocity regions on the hub and shroud side at the diffuser outlet were observed because of the secondary flow in the diffuser. From the above studies, it was clarified that the ununiform flow in the diffuser was caused by the influence of the secondary flow in the unshrouded impeller.

1. Introduction

Turbopumps used for transferring liquid oxygen or liquid hydrogen have single-stage high head. Conventionally, closed impellers have been employed for pumps. In recent years, an unshrouded impeller is being developed for high head pumps because of cost and disk friction loss reduction. Also, the flow exiting from the open impeller moves to the diffuser. There are two kinds of diffusers, vaneless diffuser and vaned diffuser. A vaned diffuser has excellent pressure conversion efficiency at the design point and is used from the viewpoint of increasing the head. However, if the vaned diffuser deviates from the design point flow rate, the performance will be degraded due to separation on the suction surface and secondary flow. Shibata et al [1]. revealed that the rotating stall occurred in the low-flow region of a vaned diffuser, and the diffuser loss increased.
However, it is known that even at the design point flow rate, pressure fluctuations occur due to the complicated flows of the alternative jet and wake flows from the impeller. Qiaorui Si et al. [2] have studied the pressure and the flow angle fluctuations when the flow from the impeller flows into the vaned diffuser from PIV measurement. They confirmed that backflow region grows until 20% of vane height. Nakagawa et al. [3] measured the flow distribution of two types of the diffuser (vaneless and vaned) by using a single-hole yaw meter. As mentioned above, there are some cases where the internal flow velocity was measured with a device such as a 3-hole pitot tube or keel probe. However, it is difficult to accurately measure the flow velocity distribution because instruments enter the flow. On the other hand, LDV measurements could be performed in a non-contact by applying laser light to the flow path.

In this paper, in order to clarify the behaviour and loss mechanisms when the flow coming out of the open impeller flows into the vaned diffuser, LDV measurements were performed and the results were compared with CFD.

2. Experimental and computational methods

2.1. Experimental Method

The equipment used in this experiment was an open impeller and diffuser performance test equipment. Figure 1 shows the overall view of the experimental device. Figure 2 shows the meridional map. The working fluid was air because the time resolution of the sensor could be increased. The air flowing in through the inlet pipe is pressurized by the diffuser pump and released to the atmosphere through the outlet pipe. The impeller of the diffuser pump was connected coaxially with the electric motor, and the rotation speed was set at 2000 [rpm] using an inverter. The flow rate was varied by adjusting the opening of the butterfly valve installed at the outlet pipe, and it was measured using the orifice installed upstream of the butterfly valve. Table 1 shows the specifications of the pump used in the experiment. The open impeller consists of 10 full blades and 10 splitter blades, and there are 13 diffuser vanes and return vanes. The outer diameter of the impeller is 400 [mm] and the blade height is 20 [mm]. The Mach number was estimated from the impeller peripheral velocity and speed of sound in air at 25 [°C].
Table 1. Pump specifications

| Specification                      | Value     |
|-----------------------------------|-----------|
| Working Fluid                     | Air       |
| Rotation Speed (min⁻¹)            | 2000      |
| Volume Flow Rate (m³/s)           | 0.0900    |
| Total Head (m)                    | 122       |
| Specific Speed (m, m³/min, min⁻¹) | 126       |
| Reynolds Number                   | 1.08×10⁶  |
| Mach Number                       | 0.121     |
| Impeller Inlet Diameter (mm)      | 215       |
| Impeller Outlet Diameter (mm)     | 400       |
| Impeller Outlet Vane Height (mm)  | 20.0      |
| Full Vane Number                  | 10        |
| Splitter Vane Number              | 10        |
| Tip Clearance (mm)                | 0.7       |

Regarding the LDV measurements, four measurement points were provided on the diffuser inlet side and five measurement points were on the outlet side as shown in Figure 3. The flow velocity was measured using the short focus 1D-LDV while moving the probe from the hub side to the shroud side. The probe was moved by a three-axis traverse device. Oil mist particles with a size of about 1 [µm] were mixed from the inlet pipe to measure the flow velocity. Table 2 shows the specifications of the LDV measuring instrument.

![Figure 3. LDV measurement points at the diffuser](image)

Table 2. LDV specifications

| Specification                      | Value     |
|-----------------------------------|-----------|
| Fringe separation (µm)            | 2.488     |
| Freq. shifting Calibration (Hz/rpm)| 238.87   |
| Probe volume distance (mm)        | 52        |
| Power in the probe volume (mW)    | 74        |
| Probe volume size (µm)            | 30        |
| Probe volume length (µm)          | 120       |

2.2. Computational Method

In this study, we performed a steady analysis using ANSYS CFX19.2, a general-purpose numerical analysis code. RANS analysis was performed at the design point flow rate, at 60% and 120% flow rates. Table 3 shows the analysis conditions. The analysis domains are the impellers and diffusers with the gap of main plate added as shown in Figure 4, and the interfaces of each domain are a mixing plane. The analysis area was a periodic boundary, and the number of grid points was about 7.11 million. 20 mesh layers were inserted into the tip clearance of the impeller. Then, based on the total...
pressure at the diffuser inlet (Pt dif in), the total pressure loss, which is a dimensionless value of the total pressure drop, was calculated using the following equation.

\[ P_t \text{ Loss} = \frac{(P_{\text{t dif in}} - P_t)}{P_{\text{t dif in}}} \quad (1) \]

### Table 3. The analysis conditions

| Code              | ANSYS CFX 19.2          |
|-------------------|-------------------------|
| Analysis Method   | SST k-\( \omega \)      |
| Total Elements    | 7.11 million            |
| Domain            | Impeller + Diffuser + Gap of main plate |
| Working Fluid     | Air at 25 \(^{\circ}\)C |
| Inlet Boundary    | Total Pressure Constant |
| Mass Flow Rate    | 0.06477(kg/s)           |
| Mass Flow Rate    | 0.10795(kg/s)           |
| Mass Flow Rate    | 0.12954(kg/s)           |
| Rotational Speed  | 2000(min \(^{-1}\))      |

3. Results and Consideration

3.1. Diffuser Inlet Flow

Experimental and analytical values of the flow velocity distribution at the inlet side for the design point flow, 120\% and 60\% flow rates are shown in Figure 5 to Figure 10. The experimental values are represented by points, and the analytical values are indicated by broken lines. The flow velocity was made dimensionless by the impeller peripheral velocity.
From the radial velocity distribution at the design point flow rate, the experimental value has a flow velocity peak around Hub to Shroud 0.7 to 0.8, whereas the computational value has a flow velocity peak around 0.6 to 0.7. Both have peaked of velocity on the Shroud side. The velocity coincided around the mainstream. However, the analysis results were lower than the experimental results near the wall on the hub and shroud side. In LDV measurements, the velocity is calculated by averaging velocity samples obtained in a gaussian distribution. However, the seeding particles hardly flow near walls, and the sample data obtained is small. If the sample data near the flow velocity 0 [m/s] decreases, the calculated measurement result will be larger than the actual flow velocity. For these reasons, the flow velocity distribution near the wall did not match. In the simulation, there is no leakage flow in the back. However, the leakage flow has occurred in the experiment. This difference causes a discordance between experimental results and computational results.

Figure 11 (a) shows the total pressure distribution and Figure 11 (b) shows the radial velocity distribution at the r-z plane of the measurement point a3. The upper part is shroud, the lower part is hub, the impeller is the right side, and the diffuser is the left side. The working fluid flows from right to left. Due to the impeller tip leakage flow and the gap of the main plate of this equipment, a vortex that flows backward at the radial velocity near the wall on the hub and shroud side of the interface between the impeller and the diffuser is generated. The flow path is narrowed by these vortices. That influence extends the radial velocity of the fluid flowing in the mainstream increases by 1.54 to 2.59 times as designed. Moreover, the backflow is induced on the hub shroud surface because the flow path was expanded before the diffuser vane.
From Figure 5 to Figure 8, experimental and analytical results were obtained in which the flow velocity decreased only at the measurement point a1 at the design point flow rate and 120% flow rate. Figure 12 and Figure 13 show the static pressure contour diagrams on the midspan at the design point flow rate and 120% flow rate. Seen the distribution of static pressure, at 100% and 120% flow rates, the static pressure increases greatly on the pressure surface side of the front edge of the diffuser. The flow velocity was smaller than the other inlet measurement points because the stagnation point is close to the measurement point a1.

However, at a flow rate of 60%, the flow velocity at a1 was same as the other measurement points on the inlet side. This is thought to be because the impeller outflow angle was reduced by the decrease of the flow rate. Then the collision of the flow on the pressure surface side of the diffuser leading edge was suppressed, and the flow velocity at the measurement point a1 was almost equal to the flow velocity near the main flow.

3.2. Diffuser Outlet Flow
Experimental and analytical values of the flow velocity distribution at the outlet side for the design point flow, 120% and 60% flow rate of design point are shown in figures 14 to 19. Similar to the graph at the inlet, the experimental values are represented by points, and the analytical values are indicated by broken lines. The flow velocity was made dimensionless by the impeller peripheral velocity.
measurement could not be performed at point b4 because one of the two lasers interfered with the diffuser.

Figure 14. Outlet Radial Velocity (design point)

Figure 15. Outlet Circumferential Velocity (design point)

Figure 16. Outlet Radial Velocity (Q/Qd=1.2)

Figure 17. Outlet Circumferential Velocity (Q/Qd=1.2)

Figure 18. Outlet Radial Velocity (Q/Qd=0.6)

Figure 19. Outlet Circumferential Velocity (Q/Qd=0.6)

Figure 20 and Figure 21 show the total pressure loss distribution at the outlet measurement point at the design point and 120% flow rate. The red region has a larger total pressure loss, while the blue region has a smaller total pressure loss. From the flow velocity distribution map and the total pressure loss contour diagram, the vicinity of the measurement point b1, b2 where the loss is small is the mainstream, and it is thought that it gradually deviates from the main flow from b3 to b5. Comparing the experimental and analytical results at the design point, the difference between the experimental and analytical results at measurement point b5 is bigger than at other measurement points in both radial and circumferential velocity. It is probable that the steady analysis could not faithfully simulate the flow because the measurement point b5 locates immediately downstream of the trailing edge of the diffuser vane. Therefore, the behaviour cannot be captured in the steady analysis, and the difference between the experimental value and the analytical value is enlarged.
From the circumferential velocity distribution with two maximum values on the hub and shroud sides at measurement points b2, b3, and b4 are seen. The experimental and analytical results showed a qualitatively close distribution. Figures 22 and 23 show the static pressure distribution and the secondary flow vector in the diffuser. Looking at the secondary flow, vortex structures could be confirmed on any surface from upstream to downstream in the diffuser. Near the surface of Hub to Shroud 0.6 to 0.7, a secondary flow is generated from the outer side of Vane1 to the inner side of Vane2. However, it is pushed back by the pressure gradient, and a secondary flow is generated in the direction opposite to the vicinity of the mid surface from the inner diameter side of Vane2 to the outer side of Vane1 at the wall on the hub and shroud side. Two vortices swirling in the opposite direction according to these flows can be confirmed. Such a vortex has a secondary flow structure that is often seen between the diffuser blades, but it could be confirmed that the boundary between the two vortices is near the surface where the radial velocity of the flow flowing into the diffuser blade is the maximum. The two secondary flow structures lead to had two maximums on the hub and the shroud side in the velocity distribution at the diffuser outlet.

Figure 24, Figure 25 shows the total pressure loss distribution between the diffuser outlet blade to blade at the design point and 120% flow rate. From these figures, it can be confirmed that the total pressure loss increases at the midspan of the vane height. These are caused by the expansion of the
separation area because the secondary flow occurred from the suction surface to the pressure surface near the midspan.

![Figure 24](image1.png)

**Figure 24.** Total Pressure Loss at the Diffuser Outlet Surface Blade to Blade (design point)

![Figure 25](image2.png)

**Figure 25.** Total Pressure Loss at the Diffuser Outlet Surface Blade to Blade (Q/Qd=1.2)

4. Conclusion

Measurements using an open impeller air test equipment and LDV and numerical calculations were carried out, and the following findings regarding the internal flow of the diffuser were obtained.

- At the outlet of the open impeller, the tip leakage flow and the gap cause the backflow of the radial velocity near the wall of hub and shroud.
- The generated backflow narrows the flow path at the impeller outlet. Therefore, the flow velocity increases in the main flow, and increases the loss due to an expansion of the flow path before the diffuser vane.
- Region where the flow velocity was low existed because of a stagnation point near the leading edge of the diffuser vane.
- The secondary flow is generated by the pressure gradient generated between the diffuser blades, and the effect causes a flow having a non-uniform velocity distribution at the diffuser outlet.
- At the diffuser outlet, reduction of the flow velocity at the midspan due to the secondary flow, causing a total pressure loss.

5. Reference

[1] Shibata A, Hiramatsu H, Komaki S and Miyagawa K 2016 Rotating Stall Characteristics in Off Design Condition of a Centrifugal Pump *Turbo-machinery* 44 pp.216-222 (in Japanese)

[2] Qiaorui Si, Patrick D, Annie C, Bayeul L, Antoine D, Olivier R and Shouqi Y 2015 An Experimental Study of the Flow Field Inside the Diffuser Passage of a Laboratory Centrifugal Pump *J. Fluids Engineering* FE-14-1322

[3] Nakagawa T, Furukawa A and Takahara H 2001 Flow Behavior Downstream of Diffuser Pump Impeller *Turbo-machinery* 29 pp.110-118 (in Japanese)

[4] Ichinose A, Kimura N, Yoshimura M, Hayashi T and Miyagawa K 2018 Investigation of Interaction Between Tip Leakage Flow Generated by Unshrouded Impeller and Diffuser Internal Flow *Proceeding of the ASME 2018 5th Joint US-European Fluids Engineering Division Summer Conference FEDSM2018-83502*. Montreal, Quebec, Canada
[5] Ichinose A, Tomoki T, Miyagawa K, Ogawa Y, Negishi H and Niiyama K 2019 Unsteady Flow in The Vaned Diffuser of a Flow Specific Speed Centrifugal Pump with an Unshrouded Impeller *Proceedings of the ASME-JSME-KSME 2019 Joint Fluids Engineering Conference, AIKFLUIDS2019* San Francisco CA USA

[6] Miyagawa K, Osada T and Aoki S 2004 Study of Internal Flow and Performance Improvement for Centrifugal Pump *22nd IAHR Symposium on Hydraulic Machinery and Systems*

6. Acknowledgements

The authors would like to thank the Waseda Research Institute for Science and Engineering (WISE) for providing support to the presented research, in context of the project: ‘High performance and high reliability research for hydraulic turbomachinery systems.’

We would like to express the deepest appreciation to Mr. Ogawa of Japan Aerospace Exploration Agency (JAXA) and Mr. Niiyama of IHI Corporation for giving an opportunity to this study.