Study on the Development of Two-Stage Centrifugal Blood Pump for Cardiopulmonary Support System

Hironori Horiguchi¹, Tomonori Tsukiya², Takeshi Nomoto¹, Toratarou Takemika¹, and Yoshinobu Tsujimoto¹

¹Graduate School of Engineering Science, Osaka University
1-3 Machikaneyama, Toyonaka, Osaka, 560-8531, Japan
²Department of Artificial Organs, National Cerebral and Cardiovascular Center
5-7-1 Fujishirodai, Suita, 565-8565

Abstract

In the cardiopulmonary support system with an ECMO (extracorporeal membrane oxygenation), a higher pump head is demanded for a blood pump. In order to realize a blood pump with higher pump head, higher anti-hemolysis and thrombosis performances, a study on the development of unprecedented multistage blood pump was conducted. In consideration of the application of the blood pump for pediatric patients, a miniature two-stage centrifugal blood pump with the impeller’s diameter of 40mm was designed and the performance was examined in experiments and computations. Some useful knowledge for a design of the blood pump with higher anti-hemolysis and thrombosis performances was obtained.

Keywords: Centrifugal Pump, Artificial Heart, Blood, Hemolysis, Thrombosis

1. Introduction

Blood pumps for the cardiopulmonary support system with an ECMO (extracorporeal membrane oxygenation) need a higher pump head of approximately 200-500mmHg due to high pressure losses in the membrane oxygenator and the cannula tubing. Existing single stage centrifugal pumps can generate the higher pump head with higher rotational speed of the impeller. However, the large shear stress due to the higher rotational speed can occur and lead to a mechanical hemolysis (destruction of red blood cells). In severe conditions of the higher pump head of 500 mmHg, the hemolysis could occur even in the Jostra RotaFlow centrifugal pump, which is considered to have high anti-hemolysis performance [1]. The cases where the hemolysis becomes a problem have been reported for pediatric patients using the ECMO [2].

In order to realize a blood pump with higher pump head, higher anti-hemolysis and thrombosis performances, a basic study on the development of unprecedented multistage blood pump was conducted. In consideration of the application of the blood pump for pediatric patients, a miniature two-stage centrifugal blood pump with the impeller’s diameter of 40mm was designed and the performance test was conducted. The simulation of the internal flow was also made and the problem to solve was extracted. Some useful knowledge for a design of the blood pump with higher anti-hemolysis and thrombosis performances was obtained and reported in the present paper.

2. Design of Pump

In the design of blood pumps, we need to consider the suppression of hemolysis and thrombosis. The type number of blood pumps is low in general and the design method of blood pumps has not been established yet. In the present study, based on the design method of general industrial centrifugal pumps, a novel two-stage centrifugal blood pump has been designed, using the knowledge of the suppression of hemolysis and thrombosis for single-stage blood pumps. The flow rate, the pump head, and the rotational speed at the design point are 3 L/min, 500 mmHg, and 4200 rpm, respectively.

The meridian cross-section of the designed blood pump are shown in Fig.1. The cross-sections from A to F in Fig.1 are shown in Fig.2. The working fluid flows into the suction volute shown in Fig.2(a), and flows through the first-stage impeller (Fig.2(c)), the return channel (Fig.2(d)), the second-stage impeller (Fig.2(f)), and the double volute (Fig.2(f)) and flows out from the pump. The suction volute (Fig.2(a)) generates a pre-swirl flow at the inlet of the impeller in the opposite direction of the rotation of the impeller for higher pump head. The double volute casing (Fig.2(f)) was adopted to decrease the radial fluid force acting on the impeller, considering the change of the present pump into a magnetically-levitated pump in the future. The casing is made of a transparent
acrylic resin for a visual observation. The black dots in Fig. 2 show the measuring points of pressure and their names are shown on the side of them.

The specification of the pump is shown in Table 1. To suppress the stagnation near the tip of blades, a semi-open type impeller was adopted. The diameter $D_t$ of the impeller is 40 mm and the pump has a small priming volume of 42 ml. The shapes of the blades of the 1st and 2nd stage impellers are the same and two-dimensional. The inlet and outlet blade angles, $\beta_1$ and $\beta_2$, are 4.7 deg and 80 deg, respectively. The inlet blade angle is small and 4 blades are given to the impeller so that the blades do not choke the flow passage at the inlet. The geometry of the blade was designed by a three-circular-arc method.

It is pointed out in some studies that there is an optimum tip clearance to prevent the hemolysis. In the study of Schima et al. [4], the tip clearance of 1.5 mm leads to the minimum amount of hemolysis in a single stage centrifugal pump with an impeller similar to the present pump impeller, although a shear velocity in the tip clearance around the periphery of the impeller in the reference [4] is low and about 27% (2300 s$^{-1}$) of the shear velocity of 8400 s$^{-1}$ in the present pump. In the studies of Miyazoe et al. [5] and
Masuzawa et al. [6], the effect of the tip clearance was examined and it was found that the shear stress in the fluid and the amount of hemolysis significantly decrease by changing the tip clearance from 0.5 mm to 1 mm. In the cases with the tip clearances of 1mm and 1.5 mm, there was no difference of the shear stress and the amount of hemolysis. Therefore, the tip clearance $C$ of 1 mm was adopted in the present study.

Assuming that each stage of the pump generates a half of total pump head, the pump in each stage has a low type number of 0.13 (specific speed is 92 (rpm, m$^3$/min, m)). Based on the design method for general industrial pumps, the blade heights at the inlet and the outlet, $h_1$ and $h_2$, are 3 mm and 1 mm, respectively. In comparison with the tip clearance of 1 mm, the blade heights are quite small and it was expected that the internal flow in the impeller was disturbed by the leakage flow though the tip clearance. Therefore, the blade heights, $h_1$ and $h_2$, were increased to 5 mm and 3 mm, respectively, and the effect of the leakage flow to the internal flow was relatively decreased. In the return channel shown in Fig.2(d), five guide vanes with the inlet blade angle of 2 deg and the outlet blade angle of 90 deg are given.

3. Experiment

Schematic of the experimental apparatus for the measurement of pump performance and pressure distributions inside the pump are shown in Fig.3. The working fluid is a water at ordinary temperatures. The water flows from the reservoir, passes through the suction tube, the pump, the discharge tube, and returns to the reservoir. To measure the pressure in the suction and discharge tubes and the inside of the pump, a water column manometer was used. For the measurement of the flow rate, the magnetic flow meter (Nihon Kohden Corporation, MFV-3200) is set in the discharge tube. The flow rate was adjusted by clamping the discharge tube or the gate valve. The pump was driven by the DC motor (Maxon motor, RE30). For the measurement of the torque, the torque meter (Ono Sokki Co., Ltd., MD-503C) was used. The rotational speed of the pump was measured by the optical detector (Ono Sokki Co., Ltd., LG-9200) and displayed by the digital tachometer (Ono Sokki Co., Ltd., TM-2110). The rotational speed of the pump was set to be 1200 rpm or 1150 rpm so that the Reynolds number $Re (= \frac{U D}{\nu})$ in the experiment using the water at ordinary temperatures was equal to be about $1.1 \times 10^5$ in the operation at the rotational speed of 4200rpm or 4000rpm with the blood assuming the density $\rho$ of 1050 kg/m$^3$, the viscosity $\mu$ of 3.25 m$^2$/s (kinetic viscosity $\nu$ of $3.1 \times 10^{-6}$ m$^2$/s).

4. Computation

For the simulation of the flow, a commercial software, ANSYS CFX-11.0 was used. The governing equations are the continuity equation and the Reynolds averaged Navier-Stokes equation. The $k-\omega$ turbulence model was used.

The computational domain consists of a rotational domain of the impeller and stationary domains of the casing, the suction and discharge tubes. The computational grid on the surface of the computational domain is shown in Fig.4. The shape of computational cell is tetrahedral in almost all of the computational domains and triangular prism near the wall. The maximum values of $y^+$ were about 14. Therefore, $\omega$ for the viscous sublayer or the blending expression of $\omega$ for the region between the viscous sublayer and the logarithmic layer was automatically selected in the computation [7, 8]. The number of cells are about 240,000, 460,000, 30,000, 30,000 for the domains of the impeller, the casing, the suction tube, and the discharge tube, respectively, and totally about 760,000. The static pressure at the inlet, the mass flow rate at the outlet, the non-slip condition on the wall were given as boundary conditions. The unsteady flow simulation was conducted with the time step of one hundredth of the rotational period. The working fluid is a water at ordinary temperatures and the rotational speed is 1200 rpm or 1150 rpm, as in the experiment.

For a pump similar to the present pump, the simulations with the computational cells of about 1.2 million or 3 million were conducted. The wall shear stress was a little larger in the case with larger number of cells. However, the distributions of the wall shear stress had no large difference qualitatively. Thus, in the view of the computational cost, the computational grid with the smaller number of cells was adopted.
5. Results and Discussions

5.1 Performance Curve

Figure 5 shows a performance curve of the prototype. The horizontal and vertical axes mean the flow rate and the pump head, respectively. The experiment was conducted with the rotational speed of 1200 rpm using the water and the results were converted into the values in the operation with the blood at 4000 rpm, considering an easy understanding in clinical practice. The demanded pump performance of 500 mmHg at 3 L/min was obtained at 4000 rpm in the experiment with the blood, although the design rotational speed was 4200 rpm. Therefore, the results were converted into the values at 4000 rpm, using the relation of \[ \frac{\Delta P_{\text{blood}}}{\Delta P_{\text{water}}} = \left( \frac{\rho_{\text{water}}}{\rho_{\text{blood}}} \right) \left( \frac{U_{1,\text{blood}}^2}{U_{1,\text{water}}^2} \right) \] derived from the condition that the values of the pressure coefficients in the operations with the water and the blood are the same because the Reynolds numbers are equal and the flow fields are hydrodynamically similar in both cases.

As a reference pump, the Jostra RotaFlow centrifugal pump (Maquet Cardiopulmonary AG) was used, which is of practical use and is considered to have high anti-hemolysis performance. The pump performance of the Jostra RotaFlow centrifugal pump is also shown in Fig.5. The results was obtained through translating the results of 1200 rpm with the water into the results of 4150 rpm with the blood. The slope of the performance curve of the prototype is steeper than that of the Jostra RotaFlow centrifugal pump in usable ranges of 1 - 5 L/min. This means that the prototype has useful characteristics that the flow rate is insensitive to the flow resistance. The results of the CFD for the prototype is also shown in Fig.5 and agree with experimental results well.

The efficiency of the prototype was also measured. As the torque generated by the friction between the shaft and the seal was different case by case in the operation using the air as a working fluid, there was variation in the efficiency obtained by substituting the torque with the air from that with the water. In the operation using the water, the torque generated by the friction between the shaft and the seal was stable. The efficiency including the friction between the shaft and the seal was 5 - 8 \% at \[ Q = Q_d (3 \text{ L/min}) \] and the maximum efficiency was 6 - 10 \% at 5 L/min.

5.2 Pressure Distribution

The pressure distribution in the pump was measured at 0.29, 0.86, 1.43, 2.00, 2.57 L/min at 1200 rpm and shown in Fig.6. These
flow rates corresponds to 1.0, 2.9, 4.8, 6.7, 8.6 L/min at the operation of 4000 rpm using the blood as a working fluid. The horizontal axis means positions shown in Fig.2. The values of In 1, Out, RC, In 2 are the averaged values of circumferential pressure distribution. The computational results are in good agreement with the experimental results except for the positions from V-5 to V-9 at higher flow rates (4.8 L/min and 6.7 L/min).

The pump head generated by the 1st and 2nd stage impellers are nearly same due to the pre-swirl flow in the suction volute and the incomplete turning in the return guide vane. The pressure recovers smoothly in the double volute (V-1 to V-4) except for the case with the higher flow rate of 8.6 L/min.

### 5.3 Anti-Hemolysis Performance

Kameneva et al. [9] examined the blood flow in a narrow tube with the inner diameter of 1 mm and showed that the amount of the hemolysis drastically increased when the shear stress on the blood was larger than 200 Pa. Therefore, the shear stress of 200 Pa...
was used as a threshold of hemolysis in the present study. The wall shear stress in the pump is shown in Fig. 7. In the operation of 1150 rpm with the water, the threshold of the shear stress for the hemolysis is about 16 Pa, based on the similarity law of

$$
\tau_{\text{water}} \cdot \rho_{\text{water}} \cdot U_{\text{water}} = \rho_{\text{blood}} \cdot U_{\text{blood}} \cdot \rho_{\text{blood}}$$

The shear stress larger than 16 Pa is observed on the casing wall near the tip of the 1st and 2nd impellers (Fig. 7(a) and (d)), the periphery of the backshroud of the 1st impeller (Fig. 7(b)), and the tongue of the double volute casing (Fig. 7(e)). The reduction of these high shear stress is necessary to suppress the hemolysis.

5.4 Anti-Thrombosis Performance

Figure 8 shows the velocity vector on the cross-section D′-D′ in Fig. 1, which is in the return channel and near the casing wall (0.5 mm from the wall). As shown in Fig. 8, the flow separates from the blade and the stagnation occurs in the center of the vortex, which can cause the thrombosis.

Figure 9 shows the flow field in the meridian cross-section around the clearance between the backshroud of the 2nd impeller and the casing. The fluid in the boundary layer of the backshroud flows outward, driven by the centrifugal force due to the rotation of the impeller. The outward flow generates the inward flow near the casing, the recirculating flow occurs in the clearance. However, as the velocity is small near the shaft, it was suggested that the 2nd impeller should have a washout hole to suppress the thrombosis. The velocity in the space around the shaft is also small and should be removed for the suppression of the thrombosis. The spaces with the possibility of the thrombosis will be removed in next prototype.

5.5 Reduction of the High Wall Shear Stress on the Casing Wall near the Blade Tip of the Impeller

It is considered that an optimal tip clearance exists for the suppression of the hemolysis. In the study of Schima et al. [4], the amount of the hemolysis became smallest in the case with the tip clearance of 1.5 mm in a single stage centrifugal pump. Therefore, we conducted the computation for the pump with the tip clearance of 1.5 mm. Figure 10 shows the wall shear stress on the casing

![Fig. 10 The effects of tip clearance and front shroud for wall shear stress on the casing wall near the 1st stage impeller. N=1150 rpm](image)

![Fig. 11 Streamline on meridian plane around the 1st stage impeller at θ=60 deg, defined in Fig.10(a)](image)
Fig. 12 Velocity vector near the casing in various types of return guide vane (return channel)

near the blade tip of the 1st impeller. Figures 10 (a) and (b) show the results for $C=1$ mm and 1.5 mm, respectively. The results shown in Figs.7(a) and 10(a) are a little different due to the difference of the return channel. It could be confirmed that the region with the shear stress higher than 16 Pa decreases due to the increase of the tip clearance, but the amount of the reduction of high shear stress region was small. Figure 10(c) shows the results in the case of the impeller with a front shroud and the clearance of $C = 1$ mm. The front shroud was quite effective for reducing the high shear stress region.

Figure 11 shows streamlines around the 1st impeller on the meridian cross-section at $\theta=60$ deg shown in Fig.10(a). The color of the streamlines means the ratio of the circumferential velocity $u_\theta$ to the tip speed $U_t$. Figures 11(a), (b), (c) show the results of $C=1$ mm, 1.5 mm, and 1 mm with the front shroud, respectively. As shown in Figs.11(a) and (b), a larger circumferential velocity occurred near the casing wall around the tip of the impeller. This is the cause of the higher wall shear stress on the casing near the tip of the impeller. In the case with the front shroud, the occurrence of the higher velocity near the casing is suppressed as shown in Fig.11(c) and this leads to the suppression of the higher shear stress.

The pump head in the cases with $C=1$ mm, 1.5 mm, and 1 mm with the front shroud are 485 mmHg, 463 mmHg, and 538 mmHg, respectively, and the pump head increased by about 11% using the front shroud. Kurokawa et al. [10] reported that the head of the pump with a low specific speed increases using the front shroud. As the increase of the pump head results in a low rotational speed of the impeller, the front shroud has an advantage for the anti-hemolysis performance.

5.6 Modification of the Return Guide Vane

As shown in Fig.8, in the return channel, the stagnation which could be the cause of the thrombosis occurred at the center of the steady vortex due to the flow separation. Therefore, the geometry of the return guide vane (return channel) was considered to suppress the stagnation. The geometries of various types of the return guide vane (return channel) were considered and the velocity field were shown in Fig.12. The vane angles of pressure surface at the inlet and the outlet are 2 deg and 90 deg, respectively. To increase the effect of the return guide vane, the number of the vanes was increased from 5 to 9 and the flow field in the case with 9 guide vanes was shown in Fig.12(a). Although the size of the vortex diminished in comparison with Fig.8, the stagnation at the center of vortex due to the flow separation occurred.

The height of the return guide vane in the present study was larger so as to be equal to the larger blade height of the impeller. For this reason, the circumferential velocity is quite larger than the radial velocity in the return channel. Therefore, longer guide vane was tried to be adopted to turn the flow gradually from the circumferential direction to the radial direction. The flow field for the longer guide vane is shown in Fig.12(b). Larger flow separation was suppressed by the longer guide vane, but the flow was not along the guide vane near the outlet of the return channel and the circumferential velocity could not be decreased well. In the above modifications, the vane with a constant thickness was adopted for easy designing of the vane but was difficult to suppress the flow separation. Then, we gave the thickness to the guide vane such that the vane covers the region of the flow separation shown in Fig.8.
The flow field for the thicker guide vane is shown in Fig.12(c) and we find the flow stagnation disappears. The design method of the return guide vane which covers the region of the flow separation is effective for the suppression of the stagnation, although the method is a trial-and-error method. In the case with the small number of the vanes such as 5, the vane is quite thick and the flow stagnation can occur near the trailing edge of the vane. Then, we designed the guide vane which covered the region of the flow separation shown in Fig.12(a) for 9 vanes. The flow field for 9 thick guide vanes is shown in Fig.12(d). Although the result is for another pump similar to the present pump, the flow separation is suppressed and the circumferential velocity is decreased well. It was found that the design method of the return guide vane which covers the region of the flow separation is effective for the suppression of the stagnation. We also found that the increase of the number of the vanes leads to the decrease of the region of the flow separation and the vane thickness and is effective for decreasing the circumferential velocity well at the outlet of the return channel.

The amplitude of the pressure fluctuation at the outlet was from -13% to +10% of the average values in the original pump and was from -7% to +4% in the pump with five thicker vanes. In both cases, the pressure fluctuations were mainly caused by the interaction of the wake of the impeller and the tongue of the double volute casing because of the smaller clearance of 2 mm between the outlet of the impeller and the tongue of the double volute casing, although the geometry of the return guide vanes affect the flow separation at the outlet to a certain degree. As the pump with 9 thicker vanes has the larger clearance of 7 mm between the outlet of the impeller and the tongue of the double volute casing, the amplitude of the pressure fluctuation at the outlet was smaller and about ±1% of the average values. Therefore, as you know, the larger clearance between the outlet of the impeller and the tongue of the volute casing is effective for the suppression of the pressure fluctuation. The larger distance is also effective for the decrease of the shear stress on the tongue, which is reported in detail in the next paper.

6. Conclusions

We carried out the research on the development of a two-stage centrifugal blood pump for the cardiopulmonary support system. The two-stage blood pump in the present study is characterized by higher discharge pressure at low peripheral speed of the impeller. The results obtained in the present study are summarized as follows.

(1) The slope of the performance curve of the prototype is larger. This means that the prototype has useful characteristics that the flow rate is insensitive to the resistance.

(2) The higher shear stress which can cause the hemolysis occurs on the casing near the tip of the impeller and the flow stagnation which can be a cause of the thrombosis occurs in the return channel if we design a centrifugal blood pump with a low type number and a semi-open impeller, mainly based on the design method for general industrial pumps.

(3) The enlargement of the tip clearance was not effective for decreasing the wall shear stress on the casing near the tip of the impeller.

(4) The front shroud of the impeller suppresses the interaction between the secondary flow and the casing and can decrease the wall shear stress on the casing near the tip of the impeller. As the front shroud increases the pump head, it can be useful to increase the anti-hemolysis performance by decreasing the rotational speed of the impeller.

(5) It was found that the design method of the return guide vane which covers the region of the flow separation is effective for the suppression of the stagnation. We also found that the increase of the number of the vanes leads to the decrease of the region of the flow separation and the vane thickness and is effective for decreasing the circumferential velocity well at the outlet of the return channel.

Improvements of the present pump and the result of the hemolysis test are reported in a next paper.

Acknowledgments

The present study was supported by JSPS Grant-in-Aid for Scientific Research (B) (Grant Number 18360094, Principal Investigator: Dr. Tomonori Tsukiya).

Nomenclature

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>C</td>
<td>Tip clearance</td>
</tr>
<tr>
<td>Dt</td>
<td>Diameter of impeller</td>
</tr>
<tr>
<td>h</td>
<td>Blade height</td>
</tr>
<tr>
<td>N</td>
<td>Rotational speed</td>
</tr>
<tr>
<td>Q</td>
<td>Volumetric flow rate</td>
</tr>
<tr>
<td>Re</td>
<td>Reynolds number (=UtDt/µ)</td>
</tr>
<tr>
<td>U</td>
<td>Flow velocity</td>
</tr>
<tr>
<td>Uc</td>
<td>Tip speed of impeller</td>
</tr>
<tr>
<td>β</td>
<td>Blade angle</td>
</tr>
<tr>
<td>Δp</td>
<td>Pressure difference</td>
</tr>
<tr>
<td>µ</td>
<td>Viscosity</td>
</tr>
<tr>
<td>ν</td>
<td>Kinetic viscosity</td>
</tr>
<tr>
<td>θ</td>
<td>Circumferential coordinate</td>
</tr>
<tr>
<td>ρ</td>
<td>Fluid density</td>
</tr>
<tr>
<td>τ</td>
<td>Shear stress</td>
</tr>
<tr>
<td>Inlet</td>
<td>Subscript</td>
</tr>
<tr>
<td>Outlet</td>
<td></td>
</tr>
<tr>
<td>blood</td>
<td></td>
</tr>
<tr>
<td>d</td>
<td>Reference</td>
</tr>
<tr>
<td>water</td>
<td></td>
</tr>
<tr>
<td>θ</td>
<td>Circumferential component</td>
</tr>
</tbody>
</table>

References


