BLOOD FLOW THROUGH CHANNELS AND CLEARANCES IN IMPLANTABLE PUMPS

Steven W. Day
Rochester Institute of Technology
Steven.Day@rit.edu

ABSTRACT
Implantable rotary blood pumps are very effective at supporting patients with heart failure. New designs demonstrate distinct advantages over their predecessor diaphragm type pumps and have generated vast interest in the medical devices community, as demonstrated by hundreds of technical publications and newer commercially available devices. In addition to mechanical design criteria, these pumps share the requirement of moving a relatively large amount of blood through a miniaturized pump without damaging the blood cells. The fluid channels within the impeller are typically 1-3 mm wide and the clearance between the blades, rotating at 2,000-10,000 rpm, and the stationary housing is approximately 100-300 µm. This paper gives examples of experimental and numerical methods to characterize the flow field, and a summary of how the flow affects blood cells and design strategies to minimize blood damage.

INTRODUCTION
An estimated 150,000 patients in the western world require heart transplantation every year, while only 4,000 (2.5%) of them receive a donor heart [1]. This lack of available donors for heart transplantation has led to a large effort since the 1960s to develop an artificial mechanical heart as an alternative to heart transplant. Within the United States, an estimated 150,000 patients die of end stage heart disease each year and as many as 35,000 of these patients may be candidates for artificial circulatory assistance [2].

Most end stage cardiac failures result from cardiac disease or tissue damage of the left ventricle, after which the left ventricle is not strong enough to deliver an adequate supply of oxygen to critical organs. In 1973, the goals of the Artificial Heart Program, a federally funded project, shifted from designing a total artificial heart to the development of a totally implantable left ventricular assist device (LVAD) [3]. The LVAD is a mechanical pump that aids, but does not replace the native heart. An LVAD can effectively relieve some strain from a native heart, which has been weakened by disease or damage, and increase blood flow supplied to the body to maintain normal physiologic function. The inlet to the LVAD is attached to the native left ventricle (Figure 1). The output of the assist pump rejoins the output of the native heart at the aorta. Blood flow from both the aortic valve and the assist pump combine and flow through the body.

Figure 1: Location of LVAD in human

Reducing the negative effects of the pump on blood is largely a fluid dynamics problem, as will be shown. The factors of blood flow that contribute to both hemolysis and thrombosis are directly related to fundamental properties of fluid flow such as shear stress and stagnation. Investigations of the internal flow field can, therefore, be used to help predict the degree of blood damage that a pump will cause. Further, the blood damage resulting from a pump may be minimized or eliminated if these features can be eliminated with geometric changes to the pump.
There are currently two types of LVADs that are clinically approved. These are diaphragm type pumps, which create a pulsed flow, and continuous speed rotary type pumps. All diaphragm type LVADs operate using a flexible membrane that acts as a pusher plate to pump blood. This diaphragm may be either electrically or pneumatically powered, but it is necessarily in contact with the blood and must flex once per heartbeat for its entire design life. The trend in LVAD design is to eliminate the flexible membrane and design a continuous flow, rotary pump. Clinical results have demonstrated that the use of a continuous (not pulsatile) rotary pump as an assist to the native heart is very effective in maintaining normal physiological conditions in the patient. It offers a huge advantage in the design of such devices because the mechanical complexity is significantly reduced as compared to pulsatile devices. Still, all of the rotary pumps in clinical use have some type of mechanical bearing that almost always leads to blood damage. Because of this, they are still not viable options for permanent implantation.

The use of a pump to maintain the patient until a viable transplant is available is termed “bridge to transplant”. All LVADs and total artificial hearts are currently used only as a bridge to transplant, not as a permanent implantation. The clinical effectiveness of LVADs has been demonstrated, but all of the currently available pumps have a limited design life because of either the damage that they cause to blood or their limited mechanical design life. Many industry and academic groups are working to develop a device that has sufficient long-term biocompatibility, performance, and reliability to be used as a permanent implant.

Virtually all pumps being developed use a non-contacting motor to rotate the impeller, but a range of strategies to support the spinning impeller are being investigated. The include mechanical bearings designed to minimize wear and damage, fluid bearings that use the blood as the bearing medium, and magnetic bearings. This paper focuses on typical flow paths for magnetically levitated rotary pumps, which are the current state of the art LVADs. They have only one moving part (the pump impeller) and this has absolutely no contact with any of the fixed parts, thus greatly reducing the regions of stagnant and high shear flow that surround a mechanical or fluid bearing.

The flow path within these rotary pumps consists of a few types of clearances, all of which have typical dimensions of less than 1mm. It’s proposed here that the study of flow through these devices may benefit from recognizing the similarity of the internal flow paths to minichannels. Additionally, the science of mini channels may be pushed by this application, where a typical channel in a rotary blood pump may include varying cross sectional area, moving walls, non-Newtonian fluid, turbulent flow, centrifugal forces, and adverse pressure gradients. The flow is critically important in these channels because high shear rates within the pump are known to contribute to potentially mortal blood clotting.

1. **Channels and clearances within rotary blood pumps**

In general, all rotary blood pumps are single stage turbo-machines with and axial, centrifugal, or in some cases mixed flow configuration. In either case, the geometry of each of these necessitates that some small channels of flow are present and play a critical role in both the desired fluid flow and the undesirable damage caused by high shear stresses. In this study, two pump geometries, representing a fairly typical centrifugal and axial flow pumps are presented.

![Figure 2: Section view of a typical centrifugal flow pump.](image)

![Figure 3: Section view of a typical axial flow pump.](image)

**Blade Passage:** Within both centrifugal and axial flow pumps, the principal fluid path is through a fluid channel referred to as the blade passage. In the unshrouded or “open” impeller arrangement, as shown in Fig 2, this channel is defined by three surfaces, the impeller hub and upper and front and back surfaces of the blades. The fourth surface that encloses the blade passage is the pump housing, which is part of the stator and does not rotate. The blade height, defining, the smallest dimension of this clearance is typically 2-3mm, qualifying it as a mini-channel.

A typical centrifugal blood pump impeller has a diameter of 50mm and rotates near 2000 rpm (33 Hz), which corresponds to a linear speed of ~5 m/sec near the outer diameter. The pressure at outer diameter, within the exit volute, is larger than the at the pump inlet, so a pressure gradient exists pushing fluids towards the inner diameter. This is opposed by the centrifugal forces of the rotating fluid which act radially outward.

A shrouded or “closed” pump impeller includes a surface on the impeller that encloses the blade passage. As a result, all walls that define the blade passage do not move relative to one another. However, this introduces a small clearance region between the top of the impeller and the pump housing that has a typical dimension of between 250 and 750 micron for a magnetically suspended pump [4], and even smaller for pumps supported on mechanical bearings. This clearance is referred to as the *front clearance* from this point forward. Tangential velocities of the impeller within this region are similar to those within the pump impeller, but a sharper gradient of velocity exists due to the difference in speeds and small clearance between the impeller and housing.

**Secondary Clearances:** A fluid channel that is similar to the front clearance exists for all centrifugal flow impellers...
between the non-bladed “bottom” of the impeller and the housing. This is referred to as the back clearance and has similar dimensions as the front clearance. In pumps where the impeller supports the impeller, this shaft usually passes through the back of the housing and obstructs the flow within the back clearance. This may or may not be a complete obstruction, depending on the design of interface of the shaft and impeller, but in some cases extensive design work has been dedicated to create an alternative flow path so that stagnant fluid is not held within the back clearance for long periods of time [5]. Flow “leakage” back through either of these clearances contributes to inefficiency. Because of this a wide variety of seals exist to reduce the flow through these clearances in typical turbomachinery. Seals are not very common in blood pumps because the stagnation of fluid in these high stress regions leads to further blood damage.

Tip gap: Most centrifugal and all axial flow pump impellers are of the “open design”. In this configuration, there exists a small clearance between the tips of the rotating impeller and the fixed housing. Relative to the reference frame of the impeller, the flow primary flow direction is parallel to the blades, but some flow always passes tangentially through this clearance. This is caused both by the boundary layer on the housing and a strong pressure gradient that moves fluid from the “pressure side” of the blade to the “suction side” of the blade. It is not clear if this passage meets all the criterion to be considered a micro-channel, but typical velocities correspond to 5 m/sec and dimensions of the gap are as small as 100 μm.

2. **Mechanical blood damage from fluid stresses**

Blood can be damaged in two principal ways by a blood pump. These are generally classified as hemolysis and thrombosis. Hemolysis is associated with the exposure of blood to high shear stresses over some length of time and it is the result of either the tearing of the red blood cell membrane or the opening of membrane pores due to mechanical stress. Red blood cells (RBC) release their contents when exposed to mechanical stress according to a theory first proposed by Rand. Rand showed that the threshold strain for cell lysis is time dependent, \( \sigma_c(t_{\text{exposure}}) \), that corresponds to the time-dependent threshold value of membrane strain rate measured by Rand. Many authors have reported values of the threshold shear stress for causing red cell damage. These are typically measured by exposing cells to known shears in a modified viscometer. A survey of the literature (Figure 4) includes a wide range of reported threshold values and both laminar and turbulent flows.

Thrombosis is a complex series of interactions involving a variety of cell types that can lead to the formation of a blood clot (thrombus). The reaction is usually initiated by the activation of platelets and their subsequent adherence of platelets and leukocytes to the endothelium (the tissue layer lining native blood vessels) or to the foreign substance of a prosthetic. This process is dependent on the material that the blood is contacting, but it is also largely a fluid dynamics effect [7]. Platelets can be activated by fluid mechanical stresses, thereby initiating the clotting cascade and can be encouraged or discouraged to adhere to surfaces by local fluid dynamic structures. This adherence is discouraged by fluid shear stress at the wall, which acts to carry platelets and other particles away from the wall. Platelet adherence and subsequent thrombus formation is encouraged by low wall shear stress and streamlines directed towards the wall, as is the case at reattachment after a separation. In the case of artificial organs, the implant constitutes this surface.

Additionally, activated platelets and small thrombus may stay suspended in the blood stream and are potentially damaging. These can adhere to biological surfaces downstream of the pump or adhere to one another to form blood-borne emboli. Emboli travel through the blood stream until they clog small peripheral arteries or are filtered out of the bloodstream by small vessels in the tissue or organs. The clogging of vital organs such as the lungs is potentially fatal. The formation and subsequent release of both attached and suspended emboli is currently the greatest challenge in the design of mechanical blood contacting devices.

It has also been shown that turbulence can cause both red cell lysis and initiation of the haemostatic reaction through platelet activation. Some confusion in the literature has resulted from the fact that the effect that momentum transfer caused by turbulent convection has on the mean flow properties is referred to as turbulent stress, or Reynolds stress. Like viscous stress, Reynolds “stress” is a second order tensor, but it must be remembered that these are not true mechanical stresses, but rather a treatment of turbulent convection. They are referred to as “shear stresses” based on their effect on the mean flow properties, not on local fluid elements the size of a blood cell. The behavior of the Reynolds stresses and the use of the word ‘stress’ in naming them have led some investigators to (mistakenly) make no distinction between Reynolds stresses and viscous stresses. In the current work, the distinction is maintained using \( \Sigma \) for viscous stresses, and \( \tau \) for Reynolds stresses.
Figure 4: Published threshold stresses for damage to red blood cells for a given exposure time. Each data point represents a separate study, for which the citation is given in the legend. The cells can withstand higher stress for a short period of time than they can for a long exposure time.

3. **Numerical Models - Computational Fluid Dynamics**

The use of Computational Fluid Dynamics to predict not only the overall pump performance, but also the distribution of shear stresses within the flow of these pumps is widespread. Due principally to limitations in computational power, the first models looked at only one clearance within the pump at a time [8, 9], but more recent models are complete three-dimensional models of all fluid passages within the pump [10, 11]. The treatment of the rotating and stationary elements within the pump is addressed using multiple frames of reference for different portions of the mesh and a handful of methods to interface the rotating and the non-rotating reference frames. All of this can be done with most commercial CFD packages, such as Fluent and ANSYS, and these are more often used than custom written CFD software.

A detailed description of the methods of computational fluid dynamics is beyond the scope of this work. It is the subject of hundreds of texts and thousands of technical publications. A brief description is contained here just to familiarize the reader with the method and its inherent complications, particularly when modeling complex geometries and turbulent flow.

Commercial CFD packages, such as TASCFlow, designed for turbomachinery, are generally used for the prediction of the flow field at a steady flow rate when designing rotary blood pumps [12]. These models have generally looked at steady flow through the pump at a flow rate that has some physiological relevance (6 liter/minute). Even for steady flow conditions, the predicted flow field is very sensitive to grid resolution and turbulence modeling [13, 14]. There is little available data showing that these numerical codes accurately predict the complicated steady flow field in these pumps.

More recently, some investigations have focused on the time dependent nature of the flow field, due both to the relative movement of the impeller and stationary blades, and to the varying flow rate generated by the beating native left ventricle. More recently, the codes allow transient boundary conditions (pressure or flow) to be applied to a complete pump (including entrance pipe and exit volute), at a significant increase in computational cost. It is even more difficult to computationally predict the time resolved flow field, so errors due to turbulence modeling and grid resolution are also expected for the pulsatile flow conditions. While comparisons of the measured and transient flow field have been made in at least one case for a diaphragm type pump [15], there is currently no experimental verification that these simulations accurately predict the time dependent local flow field within these devices.

If a computational fluid model is to be used to model pump performance and blood damage, it must not only predict qualities of the mean field, but also levels of viscous and turbulent stresses. Predicting levels of viscous stress relies on the accurate prediction of gradients of velocity in the mean field, which are generally located on solid boundaries and in the shear layer between rotating impeller and the volute. None of the models accurately predicted either of these accurately. Without using a higher-order model to solve for the Reynolds stresses explicitly, the relationship between mean flow properties, turbulent kinetic energy, and Reynolds stresses
must be accurate in order to accurately quantify levels of turbulent stresses. Given the difficulties in the accurate prediction of $k$ (shown in this chapter) and first-order relationship between Reynolds and mean stresses (which are again based on gradients in the flow that are not effectively modeled), there is no reason to believe that computed levels of Reynolds stresses are accurate.

In the current study, Particle Image Velocimetry (PIV) was used to characterize the flow within a centrifugal LVAD under steady flow conditions. The measurements presented herein consist of the mean velocity field and turbulence statistics over the range of flow rates the pump is likely to experience when implanted. Measurements are reported in the inlet, blade passage, and exit volute. Also included is a prediction of the effects that specific flow features will have on blood damage. As a result of the

**LVAD and Flow Loop.** The LVAD geometry investigated during these experiments was the second iteration of the fourth generation (CF4b) HeartQuest™ pump [21]. Pumps with identical internal flow paths have been used in animal implant tests and modeled extensively with CFD [11]. The blood-wetted flow paths of the pump are shown in a cutaway view in Fig 6. In order to accomplish PIV measurements within the LVAD, a prototype pump that allowed for optical access into the interior of the pump was built, as described elsewhere [22]. This pump was mounted on a shaft instead of suspended by magnetic bearings. This allowed control over the impeller position and construction of this test rig before the magnetic suspension system was completed. The impeller shaft had three small “teeth” located on the outer diameter that formed an interference fit with the inner diameter (ID) of the impeller. The teeth were 3mm tall (axial direction) and obstructed less than 20% of the cross-sectional area of the back clearance gap over their axial height. Prior to these experiments, computational fluid dynamics (CFD) simulations of the flow in the back clearance gap with and without these teeth were compared to confirm that the teeth did not affect the flow in the back clearance or in other regions of the pump.

**Particle Image Velocimetry (PIV)** is a technique that measures the instantaneous velocity field within an illuminated plane of the fluid field using light scattered or fluoresced from particles seeded into the fluid [23]. It differs from techniques called “flow visualization” in that PIV is quantitative. PIV measures the instantaneous two-dimensional velocity field illuminated by the laser sheet, unlike Laser Doppler Velocimetry (LDV), which is a single point measurement. PIV has recently matured to a reliable technique that is used in a wide variety of applications [24]. PIV has been demonstrated to be effective in mini and micro channels and is recently used extensively to study these flows [25]. Typically, a variant of the technique, known as micro-PIV is used wherein the entire channel is illuminated. The depth of field of the imaging system determines the plane of the measurement and spatial gradients of velocity across the channel are resolved as gradients within each acquired image pair, which is particularly useful if the flow is assumed to be two dimensional.

In the clearances within rotary pumps the flow field is three dimensional and may be resolved by traversing a measurement plane parallel to the channel surfaces and traversing this measurement plane to measure the three dimensional distribution (tomography) of velocity component lying within the plane. The channels are often large enough that the illumination is a planar light sheet, for which the thickness of the sheet defines the measurement plane. PIV
measurements have been made on plastic prototypes designed that are designed for full optical access and use mechanical bearings instead of magnetic bearings. Additionally, we have made measurements on a pump with a fully operational magnetic suspension by removing one electromagnet from the pump retrofitting the pump with custom windows to allow optical access, a laser sheet entering and a clear view for a camera to image the scattered light, to the clearance region.

![PIV setup in a shrouded centrifugal pump impeller](image)

Using a fluid with the same index of refraction as the surrounding material is known as ‘index matching’ and is critical to minimize refraction when making quantitative measurements inside small regions with complex curvature. A Kodak digital CCD camera, PIVCAM 10-30, was used for image acquisition. The camera is a 1008x1008 pixel CCD with 8-bit resolution designed for very fast acquisition of two successive frames. The laser and camera were synchronized by software and hardware sold by TSI, Inc. A dual-cavity pulsed Nd:Yag laser (Spectra Physics, Mountain View, CA) with frequency doubling crystal illuminated polymer particles (Duke Scientific, Palo Alto, CA) that are added to the flow. These particles are nominally 10 micron in diameter and are filled with a fluorescent dye that is necessary to discriminate between the light fluoresced by the particles and that scattered from surfaces of the pump [26]. The seed particles have a specific gravity of 1.05, which is a very good match to the working fluid (SG=1.05), and ensured good particle tracking [37].

The movement of the impeller blades with respect to fixed geometries such as the cut-water creates the potential for time-varying flows within the exit volute of the pump. In order to remove the effects of the rotor/stator interaction due to the blade passage, measurements within the blade passage and exit volute were made synchronously with respect to the impeller position. A pulse generator (Stanford Research Systems, Palo Alto, CA), was used to create a controlled delay between the shaft encoder and the PIV system, effectively ‘freezing’ the impeller at any given position.

In practice, the flow rate through a rotary pump varies as the heart beats, even at a constant rotational speed. This may have the physiologic advantage of eliminating regions of stagnant flow, but time-dependence further complicates both numerical models and experimental measurements of the flow. Some authors have attempted to address this issue, both with computational models [27] and empirical measurements [28, 29].

![Example measurements resulting from scanning of laser sheet](image)

**Shear stresses** on the walls are also measured experimentally in an effort to deduce flow patterns and quantify the stresses exerted by fluid onto blood cells. As an example Used a (Artificial Organs paper that I reviewed). Oil Flow [31]

**CONCLUSION**

The flow field within implantable blood pumps is critical to determining the effectiveness of the device because most of the damage caused to blood cells is the result of fluid flow. The flow paths are a series of channels that are sufficiently small to meet the criterion of mini and microchannels. This represents a very important application of minifluidics. The presence of moving boundaries, turbulence, adverse pressure gradients, and curved geometries within these channels creates a very complex flow field that may test and stretch the capabilities of current numerical and experimental methods. In short, the paper proposes that scientists from the minichannel and medical device communities recognize this common application and collaborate to advance our understanding of the topic.
ACKNOWLEDGMENTS

Financial support for some of this work has been provided by the Utah Artificial Heart Institute and the Department of Health and Human Services, National Institutes of Health, National Heart, Lung, and Blood Institute grant number R01 HL64378-01. F. Scarano for use of the PIV algorithm, and MedQuest® Products for the fabrication of the CF4b plastic prototype used for PIV measurements. SWD is currently supported in part by NIH HLBI 1RO1 HL 077085-01A1 to D.B.Olsen.

REFERENCES

