Published in final edited form as:

Artif Organs. 2011 May ; 35(5): 522–533. doi:10.1111/j.1525-1594.2010.01087.x.

CFD Design and Analysis of a Passively Suspended Tesla Pump Left Ventricular Assist Device

Richard B. Medvitz^{*}, David A. Boger^{*}, Valentin Izraelev[‡], Gerson Rosenberg[&], and Eric G. Paterson^{*,†}

^{*}Pennsylvania State University Applied Research Laboratory

[†]Pennsylvania State University Department of Mechanical and Nuclear Engineering

[‡]Advanced Bionics Incorporated (ABI)

[&]Pennsylvania State University Hershey Medical Center Department of Surgery Division of Artificial Organs

Abstract

This paper summarizes the use of computational fluid dynamics (CFD) to design a novelly suspended Tesla LVAD. Several design variants were analyzed to study the parameters affecting device performance. CFD was performed at pump speeds of 6500, 6750 and 7000 RPM and at flow rates varying from 3 to 7 liter-per-minute (LPM). The CFD showed that shortening the plates nearest the pump inlet reduced the separations formed beneath the upper plate leading edges and provided a more uniform flow distribution through the rotor gaps, both of which positively affected the device hydrodynamic performance. The final pump design was found to produce a head rise of 77 mmHg with a hydraulic efficiency of 16% at the design conditions of 6 LPM throughflow and a 6750 RPM rotation rate. To assess the device hemodynamics the strain rate fields were evaluated. The wall shear stresses demonstrated that the pump wall shear stresses were likely adequate to inhibit thrombus deposition. Finally, an integrated field hemolysis model was applied to the CFD results to assess the effects of design variation and operating conditions on the device hemolytic performance.

Keywords

TESLA Pump; LVAD; Blood Pump; CFD

1. Introduction

The first patent for a Tesla pump was filed in 1909 by Nikola Tesla and accepted in 1913 [1]. A patent for a turbine based on the same concept was subsequently filed in 1911 [2] and also accepted in 1913. Tesla's concept was for a bladeless pump which used smooth rotating discs inside a volute housing. The pump transferred energy to the fluid by taking advantage of viscous boundary layer effects occurring between closely spaced rotating parallel plates. The basic flow patterns of a Tesla pump resemble that of a centrifugal pump. The fluid enters the pump axially and a tangential velocity is imparted to the flow through the radial

Corresponding Author: Richard B. Medvitz, Research Associate, P.O. Box 30, University Park, Pa 16804-0030, Tel: (814)863-8365, Fax: (814)865-3287, rbm120@gmail.com.

Disclosure

This work was supported by the National Institute of Health through grant R01 HL081119-03.

flow rotor region. Final static pressure recovery is obtained through use of a spiraled outlet volute to convert the tangential velocity into a static pressure rise.

Despite the similarities to a centrifugal pump, a Tesla pump makes use of a different mechanism to rotate and energize the fluid. In a Tesla pump, the radial flow passage does not contain impeller blades, instead a number of closely spaced parallel plates are present. As the plates spin the fluid particles near the wall spin at the plate rotational velocity, forming a viscous boundary layer between the plates. If the gap spacing is small enough and the plate radius large enough, the upper and lower gap boundary layers will merge. Otherwise less efficient pumping will occur as the core flow through the gaps will be under-turned and contain reduced kinetic energy. For a Tesla pump the fluid will spiral outwards through the rotor region at a flow angle that is a function of RPM, gap flow rate, gap spacing and plate radius.

Though the Tesla pump has never found wide spread commercial use, there are several features that make it an attractive option for a blood pump application. The potential advantages of a Tesla pump are as follows: reduced turbulent flow stresses compared to a bladed impeller, elimination of blade to blade leakage flow, absence of cavitation conditions, gentler blood handling due to slow momentum change in the fluid, more uniform forces due to the absence of blades and significantly reduced cost due to simplicity of manufacturing and assembly [3]. Furthermore, the replacement of the mechanical bearings with the passive magnetic and hydrodynamic suspension should improve both the reliability and the hemodynamic performance.

Advanced Bionics Inc. (ABI) has previously designed a similar Tesla pump for use as a cardiopulmonary bypass pump for a commercial heart-lung machine [4–8]. This pump was significantly larger than the current pump and was designed to yield flow rates up to 10 LPM while producing in excess of 400 mmHg of pressure rise and demonstrating low levels of hemolysis. Preliminary CFD analyses to assess optimal disc numbers, disc spacings and other relevant design parameters for the current Tesla pump were previously performed by ABI.

The two blood damage phenomena potentially occurring in a mechanical heart assist device are thrombosis and hemolysis. Thrombosis refers to the formation and growth of blood clots. Hemolysis refers to damaging of the red blood cells (RBC) and is hypothesized to be a function of cellular exposure to high shear stress and the length of time over which this exposure occurs. The processes which cause thrombosis and hemolysis to occur in complex flow situations are not completely understood [9]. The general understanding is that cellular exposure to high shear stresses can lead to hemolysis [10, 11, 12] and platelet activation [13, 14] while regions of low shear stress and/or flow stagnation can be susceptible to thrombus deposition [15, 16]. Models for the prediction of thrombosis and hemolysis have appeared in the literature [9, 17, 18, 19]. These models have been based on the general understanding that blood damage is a function of cellular exposure to fluid shear stresses.

The purpose of this paper is to describe the role of CFD in the development of a novel Tesla heart pump. CFD has been used to study a series of design modifications, with the goal of maximizing pump hydrodynamic performance. Detailed analysis was used to predict performance, to guide design changes to the pump and to assess the pump hemodynamic performance.

When working within a constrained volume, the designer of a Tesla pump has several parameters which can be changed to improve the device performance. The designer can modify the number of discs, the disc shape, the gap height, the disc diameter, the rotor RPM,

the backflow channel width and the volute shape. CFD was used to study several of these design parameters as will be described.

The paper is laid out as follows. First, design specific details are given for the pump along with the design objectives. Next the CFD methodology is described including the flow solver, grid generation, turbulence modeling and boundary conditions. Next, CFD results are presented including the design modification analysis, flow field, pump performance data and device fluid shear stresses and hemolysis predictions. Finally, conclusions and future work are discussed.

2. Pump Description

The Penn State-ABI LVAD makes use of Tesla pumping technology to provide circulatory support. A detailed description of the pump, including the motor design, passive suspension system, control system, and testing results can be found in Izraelev et al. [3]. The pump will be fully implanted within the body, meaning the system (pump and motor) must fit within a limited volume. The pump makes use of a passive suspension method which eliminates the need for traditional bearings. Instead the pump is fixed in the axial direction using a passive magnetic suspension system between the stator and the rotor magnets. Radial centering of the rotor is achieved by the fluid centrifugal forces which provide a radial centering force on the rotor. An inlet pressure sensor along with an automated control system [3] adjusts the device rotation rate to maintain the proper flow at variable pump afterloads, while preventing suction events in the ventricle.

Figure 1 shows a schematic of the nominal Tesla LVAD design. The rotor is suspended within the housing and is made up of eleven equally spaced parallel discs along with upper and lower housings which contain the rotor magnets. The maximum disc diameter of 0.8 inches (2.03 cm), the disc thickness of 0.01 inches (0.025 cm) and the disc spacing of 0.02 inches (0.051 cm) were held constant throughout the CFD design iterations. The pump contains a single axial inlet and a double spiral volute which merge to a single connection at the outlet. The volute is included in the design to aid in static pressure recovery and straighten the outlet flow. The original design had a rotor length of 2.3 inches (5.8 cm); this was increased for the final design to 2.95 inches (7.5 cm) due to motor and magnet design requirements. Finally, the nominal rotor/housing gap was 0.06 inches (0.15 cm).

Design Objectives

The objectives of the Tesla LVAD are to provide full cardiac support while simultaneously demonstrating reduced blood damage in comparison to current axial and centrifugal devices. To make the pump usable in a wide range of patients it was specified that the pump should fit within a 100 cc volume and have a weight of less than 200 grams. Hydrodynamic requirements for full support require that a pressure rise of 100 mmHg be obtained at a flow rate of 6 LPM.

Design Iteration

The design process relied heavily on CFD to both predict the device hydrodynamic performance and to identify regions of potential design improvement. CFD simulations were used to guide and quantify performance gains associated with modifying the disc leading edges, the rotor inlet region and the spiral volutes. However, the process was not automated using a design optimization routine. Corresponding in vitro experiments were performed and described in Izraelev et al. [3] to evaluate the hydrodynamic and hemodynamic performance of the prototype pump.

3. CFD Methodology

The CFD is performed on the Tesla LVAD using a steady absolute frame analysis where a tangential velocity was imposed on the rotating components. The rotor was axi-symmetric with the exception of small diameter pins mechanically connecting the individual discs to each other and the rotor casing. These pins were located near the outer radius of the discs and amounted to a very small percentage of flow blockage. To simplify the analysis the connecting pins were ignored as their influence should be minimal to the device hydrodynamic performance due to their small size. The validity of this steady analysis was based on the symmetry of the rotating components, whereas non-symmetric rotating components would have caused complex interactions between the rotor and volute, thereby necessitating a complex and computationally expensive unsteady analysis with relative motion and dynamic meshing.

Multiple flow conditions were analyzed for each geometry. The flow and rotation rates were inputs to the analyses while the resultant pressure rise was the primary hydrodynamic result of the CFD. As an isolated portion of the system was analyzed and not the entire closed loop in vitro or in vivo system, varying the flow and rotation rates independently was akin to modifying the system resistance. In this manner curves of head rise versus flow rate were generated for several rotation rates.

Flow Solver

The analyses were performed using an in-house, structured, overset, finite volume solver named OVER-REL [20] which was based on the UNCLE flow solver [21, 22]. OVER-REL has been developed with an emphasis on the simulation of rotating machinery and has been applied extensively for both turbomachinery and underwater vehicle analyses. For example, Medvitz et al. [23] analyzed the single phase and cavitating performance of a centrifugal pump and compared against experimental data.

OVER-REL is a conservative, finite volume approach applied to structured multiblock grids using a time-marching, pseudo-compressibility formulation. Inviscid fluxes are formulated from the Roe-approximate Riemann solver and extended to third-order accuracy through the MUSCL scheme. Second-order accurate central differences are utilized for the viscous fluxes. A backward Euler implicit method is used to update the equations in pseudo-time. A symmetric Gauss-Seidel method is applied to solve the resulting linear system of equations. The code allows multiple-block-per-processor parallel processing using MPI for message passing. OVER-REL allows the implementation of overset grids with the overset interpolation stencils being developed using SUGGAR [24].

Grid Generation

Structured overset grids were used for the CFD analyses of the Tesla LVAD. The grids were built using the commercial grid generation package Gridgen. The nominal grid contained approximately 4 million points. A wall spacing of 5×10^{-4} inches was set along the solid walls. A short inlet section ($L/D = 2$) and a long exit section downstream of the volute merger ($L/D = 10$) were included to minimize the effects of the boundary conditions on the solution. Figure 2 shows the surface grid and an axial grid slice for the final design. Overset meshes were used to resolve the complex geometric details such as the volute inlets, the double volute merger, and the lower housing center strut.

Turbulence Modeling

Based on the Reynolds number of the flow field through the Tesla pump it is suspected that the flow will fall close to the laminar-turbulent transition regime. There are several ways in

which a Reynolds number can be defined within the pump. The fluid density and viscosity were 1060 kg/m^3 and 3.5 cP respectively. The following Reynolds numbers were computed for the base design condition of 6 LPM throughflow and 6750 RPM. A low turbulent Reynolds number of 22,100 is computed using the tip velocity ($\omega R_{\text{tip}} = 7.2 \text{ m/s}$) and

maximum disc radius ($R_{\text{tip}} = 0.01016 \text{ m}$) $\left(\frac{\rho \omega R_{\text{tip}}^2}{\mu} \right)$. A transitional pipe Reynolds number of 3,800 is obtained when using the inlet pipe diameter (0.01016 m) and averaged inlet velocity

(1.23 m/s) at 6 LPM $\left(\frac{\rho \bar{V}_{\text{inlet}} D_{\text{inlet}}}{\mu} \right)$. This leads to the conclusion that the flow entering the pump is transitional. A third Reynolds number can be defined using the gap height ($h = 0.05 \text{ cm}$) as the reference length and the magnitude of the velocity at the gap exit ($V \sim V_{\text{tip}}$), where the radial throughflow velocity is small in comparison to the plate tip velocity

$\left(\frac{\rho \omega R_{\text{tip}} h}{\mu} \right)$. This definition gives a laminar gap Reynolds number of 1,100 at the 6 LPM 6750 RPM operating condition. These Reynolds numbers show that much of the flow is transitional in nature.

Based on the low flow Reynolds number found within the pump, the CFD analyses were performed without using a turbulence model while maintaining a high level of grid resolution and tight wall spacing to facilitate wall-shear predictions. In general, the assumption of laminar flow may lead to underestimation of the fluid stresses and overestimation of the pump efficiency. To justify this approach, additional simulations were performed using both the two-equation $q-\omega$ and the one-equation Spalart-Allmaras turbulence models. These Reynolds Averaged Navier-Stokes (RANS) models resulted in less than a one-percent change in both the pump headrise and rotor torque from the laminar model.

Boundary Conditions

The following boundary conditions were used for the Tesla pump CFD analysis. A constant velocity inflow was applied two diameters upstream of the entry to the pump casing (shown in Figure 2a). A constant pressure boundary condition was applied to the outlet located 10 diameters downstream of the volute merger. No-slip zero-flux boundary conditions were applied at the stationary walls. A rotational velocity about the pump centerline of ω was applied to the pump rotating components, where ω was the pump rotation rate.

Hemolysis Modeling

The linearized integrated field method of computational hemolysis estimation developed by Garon and Farinas [17] was applied to the Tesla LVAD over the entire range of operating conditions. Garon and Farinas proposed a mathematical model to assess hemolysis by assuming the rate of hemolysis depended upon the mechanical effects of the instantaneous stress, the exposure time, and the damage history. A hyperbolic advection equation was developed by the authors to assess a linearized damage function which in turn was related back to the release of hemoglobin into the blood stream.

The power law formulation developed by Giersiepen et al. [18] relating duration of shear stress exposure to release of hemoglobin from red blood cells was used to predict the blood damage index.

$$D = \frac{\Delta Hb}{Hb} = 3.62 \times 10^{-7} \sigma_v^{2.416} t^{0.785}. \quad (1)$$

Where D is the damage function, Hb is the total amount of hemoglobin, ΔHb is the released hemoglobin, t is time and σ_v the scalar shear stress defined by:

$$\sigma_v = \frac{1}{\sqrt{3}} \sqrt{\frac{(\sigma_1 - \sigma_2)^2 + (\sigma_2 - \sigma_3)^2 + (\sigma_1 - \sigma_3)^2}{2}} \quad (2)$$

Where σ_1 , σ_2 and σ_3 are the principle shear stresses.

Garon and Farinas [17] simplified equation 1 by linearizing and differentiating in time to remove the time dependency for a steady flow field and transforming the equation from the Lagrangian to the Eulerian frame. The resultant damage index was then integrated over the device volume to yield a value for normalized index of hemolysis (NIH).

To compare against global blood damage indices obtained by in vitro hemolysis measurements, the normalized index of hemolysis (NIH) was defined as:

$$NIH = Hb \times D \times 100 \quad (3)$$

The authors noted that the method was usable for device ranking but not for predicting absolute levels of hemolysis as measured experimentally. The model was claimed to be valid so long as the quantity of plasma free hemoglobin was small in comparison to the total hemoglobin. The time-hemolysis relationship was stated to be linear on the macroscopic level of an experimental loop (though nonlinear within the device itself).

Contrary to other methods, the use of streamtraces was not relied upon, thereby eliminating a major source of uncertainty and permitting automation of the model. The method provided a framework suitable for use with any present or future power law model.

As with experimental measurements, the computational global NIH cannot be used to identify the sources of red blood cell damage, only the cumulative effect. The accuracy of the hemolysis prediction was grid dependent, as the NIH value was dependent on the accuracy of the calculation of the shear stress, particularly near the walls.

The evaluation of blood damage is an important aspect, influencing the design of mechanical blood handling equipment. The above model for hemolysis was derived from experimental measurements of steady flow fields and was applied to steady flow simulations. While useful as a guide, a computational blood damage model has yet to be developed that can reliably predict the clinical levels of hemolysis for a blood pump.

4. Results and Discussion

A Buckingham Pi analysis yields several non-dimensional parameters which are useful for

assessing and plotting pump performance. The flow coefficient $\left(C_Q = \frac{Q}{\omega D^3}\right)$, where D is the representative length scale which in this case is chosen as the rotor tip radius (0.01016 m), is a non-dimensional representation of the pump flow rate and speed. The head coefficient

$\left(C_H = \frac{gH}{\omega^2 D^2}\right)$, where gH is the pump headrise in meters, describes the device pressure rise.

The power coefficient $\left(C_P = \frac{P}{\rho \omega^3 D^5}\right)$, where P is the pump output power, non-

dimensionalizes the pump work. The efficiency $\left(\eta = \frac{C_o C_H}{C_p}\right)$ is the ratio of energy output versus energy input. Using these non-dimensional parameters allows the pump performance curves over a wide range of geometric scales and operating conditions to collapse into a single curve so long as pump similitude is maintained and permits compact presentation of performance results.

Mesh Refinement

A refined version of the baseline CFD mesh was applied to one of the design variants to verify the adequacy of the baseline CFD mesh resolution. A 2 refinement was used in all directions to refine the mesh and adjust grid point spacings. The baseline mesh contained 3,983,340 points and the refined mesh contained 11,737,533 points. Grid wall spacings were also adjusted by a 2 refinement, yielding a wall spacing of 3.5×10^{-4} inches (9E-6 m). On the refined mesh the computations gave a rotor power increase of 0.9 percent and the pump head increase of 0.6 percent. The small change in pump performance between the two meshes demonstrated that the baseline mesh resolution was adequate to provide a design assessment of the Tesla pump performance.

Design Variants

Design modifications, meant to improve the hydrodynamic performance of the Tesla LVAD, were driven by the CFD results. Design modifications focused on the plate leading edges, the rotor inlet and the volute merger region. The first two design modifications focused on improving the device performance by improving the fluid dynamics in the plate leading edge region, primarily by reducing the leading edge separations in the upper plates and the associated losses. The next series of design modifications focused on the spiral volutes, with particular emphasis on reducing the losses associated with the volute merger. The final set of modifications were necessitated by drive and suspension system considerations and was not driven by hydrodynamic considerations. In particular, for the final device, the length of the housing was increased and the thickness of backflow gaps reduced. The following gives a detailed description of the results of the plate leading edge modifications.

Figure 3 shows the disc and backflow region meshes for the original (Figure 3a) and the final designs (Figure 3b). To improve the flow through the disc gaps, the leading edges of the upper eight plates were trimmed and a diffuser was added to the upper rotor inlet to decelerate the flow. These design modifications were done to suppress the large separations observed in the upper gaps in the original design. The circumferentially averaged velocity magnitude and in-plane streamlines are shown in Figure 4. Figure 4a shows the original plate design (design 1) results and 4b the final plate design (design 3) results. The streamlines of Figure 4a show that the flow must turn more than 90 degrees to enter the upper most gaps. The inability of the fluid to make this turn results in large flow separations being formed in the upper four gaps, with the separation in the upper gap spanning nearly the entire gap height. The tapered leading edge design reduced the required fluid turning and alleviated this issue. Small separations were observed in the final design (Figure 4b) in gaps 6 to 9. However these separations were minor and provided little flow blockage.

The velocity magnitude in the upper gaps of Figure 4a become very large near the gap exit. This occurs because the upper gap through-flow is smaller due to the leading edge separations increasing the gap residence time of the fluid. This increased exposure time results in greater fluid turning occurring at the exit of the upper gaps in comparison to the lower gaps. The gap velocity magnitude of the modified design (Figure 4b) is highly uniform, suggesting the gaps workload is relatively evenly distributed.

Figure 5a shows the distribution of gap flow rate for the plate design modifications at a 6 LPM flow rate and 6750 RPM. Design 1 showed a significant reduction in the flow rate through the upper gaps, the gap flow rate increased nearly linearly moving away from the inlet. Disc design 3 shows a much more uniform mass flow distribution through the gaps. The mass flow through the upper gaps was increased and the mass flow through the lower gaps was decreased relative to the initial design. In the first gap, the massflow was doubled from the original design by eliminating the massive separation formed at the leading edge.

Figure 5b shows the total pressure rise through the gaps for the original and modified disc designs. Again the design modifications resulted in a more uniform distribution through the gaps. The original design showed very high total pressure rise through the upper gaps due to the effects of the reduced flow rate resulting in both a higher static pressure rise through the gaps and an increased tangential component of velocity at the plate outlet.

Figure 5c shows the power imparted to the fluid in each gap. The power was determined by multiplying the total pressure rise times the mass flow through each individual gap. This Figure demonstrates that despite the higher total pressure rise in the upper gaps in the original design, the work done on the fluid in the first five gaps was substantially reduced in comparison to the modified design. While the energy input to the fluid was higher in these gaps, the energy flux was lower due to the reduced mass flow rate. In total the net work done on the fluid was increased by 5 percent, from 1.96 W to 2.06 W.

The improvement to the disc inlets resulted in a 18 percent increase in headrise at the design condition of 6 LPM, 6750 RPM, increasing the pressure rise from 65.6 mmHg to 77.6 mmHg and increasing the pump efficiency from 15.5% to 17.1%. The pump head and efficiency curves for the 6750 RPM cases are shown in Figure 6. The curves show that the pump demonstrated improved performance over the entire range of operational flow rates. The head and efficiency curves also show that the greatest improvement is seen at the higher flow rates. This is expected due to the nature of the performance degradation of design 1. At the lower flow rates the leading edge separations formed in the upper gaps of design 1 decrease in size, yielding a more uniform flow distribution. As the flow rate is increased the flow blockage due to the leading edge separations increases. These separations are present in design 1 but absent in design 3, thus the improvement in performance increases as the flow rate increases. The decreased leading edge separations at the lower flow rates are observed in Figure 7 which shows the original design at 4 (Figure 7a) and 6 (Figure 7b) LPM flow rates. These same two operating points are plotted in Figure 8 which shows that the flow (Figure 8a) and power (Figure 8b) distribution is much more uniform at the lower flow rate for the initial design.

Several design iterations focusing on the volutes failed to yield significant performance gains while simultaneously fitting within the design constraints. The final design used the design 3 discs and a volute which was nominally the same as the original. Also, the house axial length was increased to satisfy motor and magnet requirements.

Figure 9 shows the non-dimensional head (Figure 9a) and efficiency (Figure 9b) curves for the final design. Both plots collapse to a single curve for the different rotation rates when plotted non-dimensionally, suggesting similitude is maintained and Reynolds number effects were minor. The final design produced 77 mmHg of pressure rise at a hydraulic efficiency of 16% at the 6 LPM 6750 RPM design condition.

Hemodynamic Performance Estimation

The current understanding of blood damage in artificial devices implies that cellular trauma is a function of the duration of exposure to mechanical stresses. To study the hemodynamic

performance of the device the computational shear stresses were interrogated. Scalar strain rates and shear stresses were computed using equation 2.

Figure 10 shows locations of surface fluid shear rate of less than 1000 s^{-1} for the final design operating at 6 LPM and 6750 RPM. Based on the work of Hubbell and McIntire [15] and Balasubramanian and Slack [16] this level was assumed to be adequate to suppress thrombus deposition. The Figure shows that the majority of the flow field maintains high levels of fluid strain rate which would suppress the level of thrombus deposition on the non-biological surfaces of the device. The most likely thrombus nucleation sights are on the lower center strut and at the volute merger.

Figure 11 shows the shear stresses occurring on the disc upper surfaces, the disc lower surfaces and the rotor. The majority of the fluid shear stresses on the rotor are below 300 Pa. The highest shear stresses are found on the upper side of the disc leading edges as can be observed in Figure 11a. The peak shear stresses occurring in the final device design fall within the range of 300 to 500 Pa depending on the operating conditions. These high stress regions are small which should reduce the cellular exposure time, limiting the level of hemolysis. The values of shear stress found within the device are high enough to warrant in vitro platelet activation and hemolysis investigations as CFD models have not matured to the point where in vivo levels of blood damage can be predicted with sufficient certainty.

Despite the limitations of computational hemolysis modeling, i.e. the models are typically developed based on simple couette flow experiments, LVAD's likely experience shear and time scales well outside of the range over which the models were developed, and the models have thus far been unable to predict measured values of NIH; the method developed by Garon and Farinas [17] and Farinas et. al. [19] was applied to the Tesla pump with the goal of predicting the proper hemolysis trends occurring within the device, while acknowledging that the absolute levels of the NIH predicted may not match measured levels. Figure 12 plots the results of the hemolysis analysis. Figure 12a shows the median flow field shear stress versus flow coefficient over the range of flow conditions for the design pump. As expected, the trend in the data shows that shear stress increases with both flow rate and rotation rate.

Figure 12b shows that the level of hemolysis increases with increased rotation rate. This is expected since the increased rotational rate results in higher wall shear rates. The second trend observed is that hemolysis decreases with increased flow rate. This observation is less intuitive since it conflicts with the observed increase in fluid shear with increased flow rate. However, hemolysis (as shown by equation 1) is a function of both fluid shear and exposure time. At increasing flow rates the decrease in red blood cell exposure time has a greater effect than the relatively minor increase in fluid shear stress, resulting in decreased hemolysis predictions.

5. Conclusions

Significant hydrodynamic performance gains in the Tesla LVAD were achieved by modifying the disc inlet region to reduce leading edge flow separations. The design modifications yielded a much more uniform distribution of mass flux and power between the gaps and resulted in a more efficient device.

Hemodynamically, the wall-strain rate levels within the device along with the absence of flow stagnation zones suggested that surface thrombus deposition should be inhibited within the device. Based on the CFD analyses the device shear stress levels, with peak shears of 300–500 Pa, are high enough that both platelet activation and hemolysis may occur and should be studied further experimentally. The cellular exposure time to shear stresses also needs factored into the assessment of both platelet activation and hemolysis and the

integrated field method applied in this paper was an attempt to account for both shear stress and exposure time as it relates to hemolysis. A published hemolysis model was applied to assess the hemolysis trends. While the model is not intended to quantitatively reproduce measured hemolysis results, the model NIH is used as an index to qualitatively compare the hemolytic performance of design variants and operating conditions. The model suggests that increased rotation rate and decreased flow rate will in both cases result in increased levels of hemolysis.

Future work should be aimed at validation of the design CFD simulations against in vitro performance data, where a shafted rotor may be needed to guarantee experimental rotor centering. In addition, in vitro hemolysis measurements would allow refinement of the computational hemolysis model.

Also, we acknowledge that some effects were not modeled in the simulations and should be incorporated. The inclusion of magnetic forces and rotor stability would greatly enhance the utility of the CFD in aiding the design of this pump. In vitro experiments have shown that the rotor experiences an off-center axial displacement when operating, whereas the CFD was of the idealized centered case. The magnetic forces are known and should be incorporated into an unsteady CFD analysis to determine rotor stability.

Acknowledgments

This work was supported by the National Institute of Health through grant R01 HL081119-03.

References

1. Tesla, N. Fluid Propulsion. U.S. Patent 1,061,206. 1913.
2. Tesla, N. Turbine. U.S. Patent 1,061,206. 1913.
3. Izraelev V, Weiss B, Fritz B, Newswanger R, Paterson E, Snyder A, Medvitz R, Cysyk J, Pae W, Hicks D, Lukic B, Rosenberg G. A Passively-Suspended Tesla Pump Left Ventricular Assist Device. *ASAIO Journal*. 2009; 55:556–561. [PubMed: 19770799]
4. Izraelev, V. Bearing and Seal-Free Blood Pumps. U.S. Patent 5,685,700. 1997.
5. Izraelev, V. Blood Pump Having Radical Vanes with Enclosed Magnetic Drive Components. U.S. Patent 5,924,848. 1999.
6. Izraelev, V. Blood Pump Having Rotor with Internal Bore for Fluid Flows. U.S. Patent 5,938,412. 1999.
7. Izraelev, V. Improved Rotor for Blood Pump. U.S. Patent 6,206,659. 2001.
8. Izraelev, V. Pump Assembly with Bearing and Seal-Free Reusable Impeller for Fragile and Aggressive Fluids. U.S. Patent 6,547,539. 2003.
9. Arora D, Behr M, Pasquali M. A Tensor Based Measure for Estimating Blood Damage. *Artificial Organs*. 2004; 28(11):1002–1015. [PubMed: 15504116]
10. Behr M, Arora D, Coronado O, Pasquali M. Models and Finite Element Techniques for Blood Flow Simulation. *International Journal for Computational Fluid Dynamics*. 2006; 20:175–181.
11. Deutsch S, Tarbell JM, Manning KB, Rosenberg G, Fontaine AA. Experimental Fluid Mechanics of Pulsatile Artificial Blood Pumps. *Annual Review of Fluid Mechanics*. 2006; 38:65–86.
12. Baldwin JT, Deutsch S, Geselowitz DB, Tarbell JM. LDA Measurements of Mean Velocity and Reynolds Stress Fields within an Artificial Heart Ventricle. *Journal of Biomechanical Engineering*. 1994; 116:190–200. [PubMed: 8078326]
13. Bluestein D. Research Approaches for Studying Flow-Induced Thromboembolic Complications in Blood Recirculating Devices. *Expert Rev Medical Devices*. 2004; 1(1):65–80.
14. McIntire, LV. Report of the National Heart, Lung, and Blood Institute Working Group. U. S. Department of Health and Human Services, Public Health Service; Bethesda, MD: 1985. Guidelines for Blood-Material Interactions. NIH publication 85–2185

15. Hubbell JA, McIntire LV. Visualization and Analysis of Mural Thrombogenesis on Collagen, Polyurethane, and Nylon. *Biomaterials*. 1986; 7:354–363. [PubMed: 3778995]
16. Balasubramanian V, Slack SM. The Effect of Fluid Shear and Co-Adsorbed Proteins on the Stability of Immobilized Fibrinogen and Subsequent Platelet Interactions. *J Biomater Sci Polymer Edn*. 2002; 13(5):543–561.
17. Garon A, Farinas MI. Fast Three-Dimensional Numerical Hemolysis Approximation. *Artificial Organs*. 2004; 28(11):1016–1025. [PubMed: 15504117]
18. Giersiepen M, Wurzinger LJ, Opitz R, Reul H. Estimation of Shear Stress-Related Blood Damage in Heart Valve Prosthesis: In Vitro Comparison of 25 Aortic Valves. *International Journal of Artificial Organs*. 1990; 13:300–306. [PubMed: 2365485]
19. Farinas MI, Garon A, Lacasse D, N'dri D. Asymptotically Consistent Numerical Approximation of Hemolysis. *Journal of Biomechanical Engineering*. 2006; 128:688–696. [PubMed: 16995755]
20. Boger, DA.; Dreyer, JJ. Prediction of Hydrodynamic Forces and Moments for Underwater Vehicles Using Overset Grids. 44th AIAA Aerospace Science Meeting and Exhibit, AIAA-2006-1138; Reno, Nevada. January 9 – 12, 2006; 2006.
21. Taylor, LK. PhD thesis. Mississippi State University; 1991. Unsteady three-dimensional incompressible algorithm based on artificial compressibility.
22. Taylor, LK. AIAA, Paper No 91-1650. 1991. Unsteady Three-Dimensional Incompressible Euler and Navier-Stokes Solver for Stationary and Dynamic Grids.
23. Medvitz RB, Kunz RF, Boger DA, Lindau JW, Yocum AM, PLL. Performance Analysis of Cavitating Flow in Centrifugal Pumps Using Multiphase CFD. *ASME Journal of Fluids Engineering*. 2002; 124:377–383.
24. Noack, RW. SUGGAR: a General Capability for Moving Body Overset Grid Assembly. 17th AIAA Computational Fluid Dynamics Conference; June 6–9 2006; Toronto, Ontario, Canada. 2006.

Figure 1.
Schematic showing the nominal motor and pump design for the Tesla LVAD.

Figure 2. Overset computational mesh for the Tesla LVAD showing (a) the outer casing and (b) a slice through the midplane of the rotor for the original design.

Figure 3. Computational meshes showing the disk and rotor design modifications going from (a) the original to (b) the final design.

Figure 4. Circumferentially averaged velocity magnitude contours and in-plane streamlines for (a) design modification 1 and (b) design modification 2.

Figure 5.

Distribution of disc (a) gap flow rates, (b) gap total pressure rise and (c) gap power for the initial and final disc leading edge designs. The data shown is for the 6 LPM 6750 RPM case. It should be noted that gap 1 is closest to the inlet and gap 12 furthest.

Figure 6.
(a) Pump head and (b) efficiency curves for the initial and final design modifications focused on the disc inlet region at 6750 RPM.

Figure 7. Circumferentially averaged velocity magnitude contours and in-plane streamlines for design modification 1 at (a) 4.0 LPM 6750 RPM and (b) 6.0 LPM 6750 RPM.

Figure 8. Distribution of disc (a) gap flow rates and (b) power input for the original disc design at 4.0 LPM and 6.0 LPM.

Figure 9. Non-dimensional pump performance of the final Tesla pump at three different RPMs. The figures show (a) head coefficient versus flow coefficient and (b) efficiency versus flow coefficient.

Figure 10. Regions of surface fluid strain rates below 1000 s^{-1} for the final design of record at 6 LPM and 6750 RPM.

Figure 11. Wall shear stresses on the discs and rotor for the final design of record at 6 LPM and 6750 RPM, (a) the upper disc surfaces (b) the lower disc surfaces and (c) the entire rotor.

Figure 12. Final design (a) median flow field shear stress (b) and computed NIH versus pump flow coefficient.