Published in final edited form as:

Artif Organs. 2013 April ; 37(4): 380–389. doi:10.1111/j.1525-1594.2012.01576.x.

Comparison of fluid dynamic numerical models for a clinical ventricular assist device and experimental validation

Jiafeng Zhang, Pei Zhang, Katharine H. Fraser*, Bartley P. Griffith, and Zhongjun J. Wu

Artificial Organs Laboratory, Department of Surgery, University of Maryland School of Medicine, Baltimore, MD, 21201 USA

*Department Of Bioengineering, Imperial College London, London, UK

Abstract

With the recent advances in computer technology, computational fluid dynamics (CFD) has become an important tool to design and improve blood contacting artificial organs, and to study the device-induced blood damage. Commercial CFD software packages are readily available, and multiple CFD models are provided by CFD software developers. However, the best approach of using CFD effectively to characterize fluid flow and to predict blood damage in these medical devices remains debatable. This study aimed to compare these CFD models and provide useful information on the accuracy of each model in modeling blood flow in circulatory assist devices. The laminar and five turbulence models (Spalart-Allmaras, k- ϵ (k-epsilon), k- ω (k-omega), SST (Menter's Shear Stress Transport), and Reynolds Stress) were implemented to predict blood flow in a clinically used circulatory assist device, CentriMag® centrifugal blood pump (Thoratec, MA). In parallel, a transparent replica of the CentriMag® pump was constructed and selected views of the flow fields were measured with digital particle image velocimetry (DPIV). CFD results were compared with the DPIV experimental results. Compared with the experiment, all the selected CFD models predicted the flow pattern fairly well except the area of the outlet. However, quantitatively, the laminar model results were the most deviated from the experimental data. On the other hand, k- ϵ RNG models and Reynolds Stress model are the most accurate. In conclusion, for the circulatory assist devices, turbulence models provide more accurate results than laminar model. Among the selected turbulence models, k- ϵ and Reynolds Stress Method models are recommended.

Keywords

computational fluid dynamics; circulatory assist device; turbulence model; blood pump; blood flow

Introduction

Heart-related diseases are the leading causes of death in the United States; approximately 25% of deaths (595,444) in 2010 were a result of heart-related diseases (1). Over 27 million

Address correspondence and reprint requests to Prof. Zhongjun J. Wu, Artificial Organs Laboratory, Department of Surgery, University of Maryland School of Medicine, 10 S. Pine Street, Baltimore, MD 20201, USA. zwu@smail.umaryland.edu. Telephone: (410) 706-7715. Fax: (410) 706-0311.

The Authors state that this manuscript has not been published previously and it is not currently being assessed for publication by any journal other than the International Journal of Artificial Organs.

Each Author has contributed substantially to the research, preparation and production of the paper and approves of its submission to the journal.

adults in the United States were diagnosed with a heart disease in 2010, which accounted for approximately 12% of the population in that year (2). The estimated direct and indirect cost of heart diseases in the United States is \$564.9 billion for 2015 and \$1117.6 billion for 2030 (3). Among all heart-related diseases in 2010, approximately 6.6 million American adults had heart failure (HF), and a 25% increase in prevalence of HF by 2030 is projected (3). The mortality rate of HF is high; approximately 50% of people die within 5 years after being diagnosed with HF. The estimated direct and indirect cost of heart failure in the United States is \$44.6 billion for 2015 and \$97 billion for 2030 (3). Despite many available therapies, a heart transplant is the only option for some patients with end-stage HF. However, only a limited number of people receive a heart transplant due to the scarce supply of donor hearts. For example, only 2,322 patients in the United States were able to receive heart transplants in 2011, while over 3,000 are still waiting for a transplant (4). In 2008, the mortality rate for patients on the waiting list was 170 per 1000 patient-years (4). To address the need of the patients with end-stage HF, a wide variety of mechanical circulatory support devices have been developed. Among these mechanical circulatory support devices, ventricular assist devices (VADS) are widely used to replace or assist the function of one or more chambers of the failing heart. Detailed reviews of the ventricular assist devices (VADS) could be found in the review by Norman (5), Behbahani et al. (6), and Fraser et al. (7).

Computational modeling has been used for decades in engineering for analyzing or predicting performance and function of aircrafts and hydrodynamic machinery in the design and development process. In the recent years, the increase of computational resources has resulted in frequent use of computational fluid dynamics (CFD) for evaluating existing blood-wetted biomedical devices or new design concepts (7). CFD has often been used to determine the characteristics of these devices in terms of flow rate, pressure, velocity field, shear stress field, and blood damage potential (8) (9) (10) (11) (12) (13) (14). Compared with the experimental techniques, CFD provides a relatively efficient way to study and improve device performance by avoiding producing expensive prototypes. Details of flow field with complex geometry and various boundary conditions where experimental data are usually difficult to obtain, and immeasurable data such as stress, strain and strain rate etc, can be derived for quantitative comparison. Various CFD models, i.e. laminar vs. turbulence, can be utilized in the CFD simulation. Commonly employed turbulence models in engineering applications include Spalart-Allmaras, $k-\epsilon$ (k-epsilon), $k-\omega$ (k-omega), SST (Menter's Shear Stress Transport), and Reynolds Stress Method. However, the best CFD approach of how to effectively use CFD to characterize fluid flow and to predict blood damage in these medical devices remains debatable. In essence, the goal of this study is to ascertain the best CFD model for the rotating blood pumps.

A clinically used circulatory assist device, CentriMag[®] centrifugal blood pump (Thoratec, MA), was used in the study. Blood flow in the CentriMag[®] blood pump was modeled with different CFD models, including Laminar and six turbulence models. To evaluate these CFD models, the velocity fields in selected view planes were measured with the digital particle image velocimetry (DPIV). The CFD results were compared with the experimentally measured velocity fields.

Materials and Methods

CentriMag[®] Device

The CentriMag[®] blood pump (Thoratec, MA) is an extracorporeal circulatory support device providing hemodynamic stabilization in patients in need of cardiopulmonary support. It was designed based on a centrifugal pump with magnetically levitating impeller (see Figure 1). The CentriMag[®] blood pump is capable of operating over a range of rotating

speeds up to 5000 rpm and generating the flow rate up to 9.99 L/min under normal physiologic pressure and conditions. The internal priming volume of the pump is 42 ml and the radius of the impeller is 21.2 mm. In this paper, we chose the rotating speed at 3000 RPM with the flow rates of 1 L/min, 3 L/min, 5 L/min, and 7 L/min. The Reynolds numbers (Re_{in}), based on the inlet diameter, are 678, 2035, 3392, and 4748 respectively.

Experimental Setup

The experiment was carried out in a steady flow mock loop consisting of a reservoir, a transparent acrylic replica of the CentriMag[®] pump, tubing and a resistor. Figure 2 displays the experimental configuration. De-ionized water and glycerin were mixed by volume of 4:1 to produce a solution with a similar kinematic viscosity of blood in large vessels. Sodium iodine (NaI) was added to the solution to adjust the refraction index to 1.491 to match with that of the acrylic model. The kinematic viscosity of the solution is 3.5 cSt (± 0.2 cSt). The solution viscosity was measured using Cannon-Manning semi-micro viscometer (model M187/75, Cannon Instrument CO., State College, PA). The Cannon-Manning semi-micro viscometer was calibrated with the pure water at the room temperature before measuring the solution. The measured viscosity of the pure water is within 1% difference with the standard. The flow rate through the hydraulic loop was measured by a Transonic T410 Transit-Time Tubing Flow meter.

The 2D DPIV measurements were carried out with a custom-built digital particle image velocimetry system. A Solo Nd: YAG laser (New Wave Research, CA) and the Berkeley BNC 500 pulse generator were combined to produce double pulsing laser light sheets sequence with pre-defined time delay between two pulses. A digital CCD camera (Megplus ES 1.0, Red Lake, San Diego, CA) connected to an EPIX frame grabber (PIXCI D, EPIX Inc, Buffalo Grove, IL) hosted in a PC was used to capture and store 32 pairs of images in sequence with assorted optics. The camera CCD chip had a resolution of 1008×1018 pixels and a frame rate of 30 Hz. A laser sheet with the thickness less than 1 mm was obtained by a cylindrical lens with a focal length of 25 mm and a spherical lens with a focal length of 50 mm. The replica of the centrifugal pump was positioned at the thinnest part of the laser sheet. The time delay between two subsequent laser pulses was from 20 μ s to 80 μ s, according to the maximum flow velocity magnitude. The solution was seeded with 10 μ m red fluorescent polymer micro-spherical particles (Duke Scientific Corp, Palo Alto, CA). Illumination by the laser was at 532 nm and the particles fluoresced at 610 nm, an orange sharp cut filter with cut-off of approximately 560 nm (Hoya, Santa Clara, CA) was used to block the scattered laser light.

The impeller rotated at the speed of 3000 RPM while the flow rates were varied from 1 L/min to 7 L/min by adjusting the resistor in the flow loop. The acquired images were analyzed using the fuzzy logic enhanced DPIV Processing software developed by Wernet (1999) at US NASA Glenn Research Center. Cross-correlation was performed by the DPIV processing software to determine the displacement of the particles within 50% overlap of interrogation window of 16×16 pixels in size. The estimated error in measured velocity is approximately between 0.02 and 0.1 m/s.

Numerical Simulation Setup

The geometry of the CentriMag[®] pump was taken from the computer aided design (CAD) files. Commercial software (Ansys) was used to build up the model of the blood contacting surface and fluid flow domain. The flow domain was extracted using Ansys DesignModeler and was meshed using Ansys Meshing. The corresponding fluid flow was calculated using the commercial, finite volume software Fluent 12.0 (Ansys Inc). Motion of the impellers was incorporated using the multiple reference frame (MRF) approach. MRF is a steady state

approximation of the actual flow in which the flow domain could be separated and assigned different rotating and/or translational speeds. The impeller was enclosed in a region with axial symmetry. In this paper, the fluid in the region surrounding the impeller is calculated in a frame of reference moving with the same velocity as the impeller, and flow in the outer region is calculated in a stationary reference frame. At the interface between the cell zones, a local reference frame transformation is performed to enable the flow variables in one zone to be used to calculate fluxes at the boundary of the adjacent zone. Re_{in} is between 600 and 5000 when flow rate increases from 1 L/min to 7 L/min. The laminar model and six turbulence models were implemented. Second order upwind discretization was used for all the equations and the SIMPLE scheme was used for pressure-velocity coupling. Simulation is considered convergent when the monitored residuals are smaller than 10^{-3} . Blood was treated as an incompressible Newtonian fluid with a viscosity of 3.6 cp (kinematic viscosity of 3.43 cSt) and a density of 1050 kg/m³. Boundary conditions were a) a uniform flow at the inlet entrance, and b) uniform pressure at the outlet exit. Gravity is not considered in the simulation.

In order to decrease the computational time and memory requirement, a hybrid mesh model is used which uses both hexahedral and tetrahedral elements. The hexahedral elements were used for the inlet and outlet, and the remaining portion was meshed with tetrahedral cells (Figure 3). The mesh independent analysis was carried out to ensure that the CFD simulation results were independent of further mesh refinement. Three meshes with the same structure but different element sizes were generated. The coarse, medium and fine meshes have approximately 2.3 million, 3.1 million and 4.3 million elements respectively. The element refinement ratio (r) was calculated as $(\text{mesh volume finer}/\text{mesh volume coarser})^{1/3}$. The discretization errors were then analyzed according to the principles outlined by Roache (15). Figure 4 showed the mean discretization error of the coarse, medium and fine meshes. As mesh was refined and the element size decreased, the average discretization error decreased. The mesh with the highest resolution was chosen to compare the selected CFD models.

Seven CFD models were compared: Laminar, k- ϵ renormalization group theory (RNG) with enhanced wall treatment, k- ϵ RNG without enhanced wall treatment, k- ω SST, Spalart-Allmaras, Transition SST, and Reynolds Stress. Standard k- ϵ and k- ω models were not included, and the reason will be discussed in the Discussion section. In order to keep the study simple, all the parameters of the CFD models were the default values by the commercial software Fluent (values refer to the Ansys Fluent manual).

Data Analysis

Previous research showed that a large number of image pairs for DPIV are required for turbulent flows (16) (17). For the CentriMag[®] pump in the DPIV experiment, we increased the number of collected image pairs from 64 to 320, and we monitored the average velocity magnitude differences until the average differences did not change with the number of image pairs any more. Specifically, the average velocity magnitude difference is less than 1% when the number of image pairs was larger than 128. All the DPIV data presented in this paper were averages of 160 pairs of images. Furthermore, previous research showed that a low pixel resolution may result in a bias error due to the peak locking (18) (19) (20). In order to confirm that the pixel resolution is high enough, we also zoomed in at a smaller flow field so that the higher resolution (18 $\mu\text{m}/\text{pixel}$) images were achieved. The velocity field analyzed from the higher resolution images were compared with the whole pump flow field with the lower resolution (70 $\mu\text{m}/\text{pixel}$), and the difference was negligible. Thus, we presented the DPIV data of the whole flow field in this paper.

We chose the average relative error of the velocity field to quantify the differences between the CFD simulations and the DPIV experiment. The average relative error of the velocity field was calculated as the following:

$$error = \frac{1}{n} \frac{\sum_{i=1}^n |U_i^{exp} - U_i^{num}|}{average(U_i^{exp})} \quad (1)$$

where n is the total number of points in the 2-D velocity profile, superscripts *num* and *exp* refer to the numerical and PIV experimental velocity value, U refer to the 2-D velocity magnitude, and i is one discrete point. Only the velocity magnitudes were used in the error analysis because of the nature of the rotating flow. The velocity component in the x (or y) direction may vary from 0 to a few meters per second at the different locations, which made it very difficult to measure accurately in PIV experiments. Specifically, it required long times between image pairs where the velocity in x (or y) direction is small, but particles in the area with high x (or y) velocity component usually moved out of the image frame.

Results

Pressure Difference between the Inlet and Outlet

The pressure difference ($\Delta P = P_{out} - P_{in}$) between the inlet and outlet (see Figure 1b for the location of the inlet and outlet) were compared between the experimental measured and predicted from the selected CFD models (Figure 5). All pressure difference data were normalized by the average velocity at the inlet and the density of the fluid,

$$\bar{u}_{in} = \frac{Q}{\pi r_{in}^2} \quad (2)$$

$$\Delta P_{norm} = \frac{2\Delta P}{\rho \bar{u}_{in}^2} \quad (3)$$

where $\bar{u}_{in}$ is the mean velocity at the inlet calculated from the flow rate and the inlet diameter.

Due to the variance in simulated pressure, all CFD pressure results were the data averaged from the 1000 simulation steps after the simulations converged. All the CFD models fairly well predicted the pressure difference between the inlet and outlet. Reynolds Stress model and k- ϵ RNG models under-predicted the pressure difference for all Re_{in} , while other models under-predicted the pressure difference for the smaller Re_{in} (i.e. smaller than 3000) but over-predicted for the larger Re_{in} (i.e. larger than 4000).

Velocity Profile along Radial Cut

Comparison of the velocity magnitude at the radius of 25.3 mm at the rotational speed of 3000 RPM and flow rates 1, 3, 5 and 7 L/min is presented in Figure 6. The x-axis is the angle in radians, and 0 is shown as point A in the insert of Figure 6; the y-axis is the velocity magnitude in m/s. The lack of data in the area around $3\pi/2$ was because the tubing connected to the inlet of the CentriMag® pump blocked the viewing area. As we expected, the velocities at a fixed radius are approximately constant. If the fluid rotate as a rigid body, the velocity at $r = 25.3$ mm would be approximately 7.9 m/s given the rotating speed of the impeller at 3000 RPM. Since the fluid in the CentriMag® pump is viscous and radially moves outward, the velocity is smaller, approximately 4 m/s.

The measured velocity magnitude showed a notable fluctuation in the area between $\pi/4$ and $\pi/2$ where the outlet (outlet location refers to Figure 1b) is located; this is the area where flow separates and exits the pump through the outlet (see the rectangle area in the insert of Figure 6). Overall, all numerical simulations captured the velocity magnitude fairly well. However, all the CFD models predicted relatively large velocity fluctuations compared with the measured data, especially in the area between the $\pi/4$ and the $\pi/2$ where the outlet is located. This probably suggests CFD models with the ability to well capture the flow transition.

Average Error of the Whole Velocity Field

The global mean error of the velocity magnitude of the 2D plane was plotted as a function of Re_{in} against selected CFD models in Figure 7. The k- ϵ RNG models and Reynolds Stress model most accurately predicted the velocities for all Re_{in} , while the velocity magnitude predicted with the Laminar model and the Spalart-Allmaras model had much larger error at all Re_{in} . The accuracy of the k- ω SST model fell in the middle at all Re_{in} . Figure 7 showed that the k- ϵ RNG model with the enhanced wall treatment (dash line with X symbol) had a smaller error than the k- ϵ model with the standard wall treatment (dash line with triangular symbol), which indicates that an appropriate wall function could improve the accuracy of the simulation without significantly increasing the computational cost.

The flow field near and in the outlet contributed the most to the error. In the area away from the outlet, the simulated velocities agreed with the experimental results better, while the velocities near the outlet (for the location of the outlet, see Figure 1b) showed relatively large deviation (see Figure 8), which probably is the main reason why the laminar model gave the largest error (Figure 8).

Discussion

The purpose of this study is to 1) establish a method to quantitatively validate the CFD modeling of the VADs; 2) establish the best practice for applying CFD modeling on VADs development and related blood damage studies. This paper presented results of point to point velocity field validation inside a VAD and compared the 7 commonly utilized CFD models.

Many previous CFD studies (21) (22) (23) (24) (25) only quantitatively compared the head pressures measured at the inlet and outlet. However, comparing the head pressures provided no validation of the flow field inside the pump since one pressure could correspond to multiple possible velocity fields. In this paper, we were able to validate the simulated velocity field inside the pump with the experimental measurement. Interestingly, all CFD models estimated the pressure fairly well. However, they didn't estimate the velocity field equally well. Moreover, contrary to the expectation, most CFD models gave a worse prediction for the smaller Re_{in} (i.e. <1000) although the flow at the inlet is considered as laminar flow. Some previous studies (26) (27) chose the Laminar model to simulate VADs based on Re_{in} . However, this paper showed that a more complicated model should be utilized even when the inlet flow is laminar. The reasons behind are multiple: 1) part of the flow inside the pump are medium to high turbulence. The turbulence intensity calculated from the DPIV data is larger than 1% around the impeller and larger than 5% in the outlet where eddies exist; 2) flow near the outlet is not steady when the inlet Re_{in} is small (or flow rate is low). We observed multiple small recirculation zones at the outlet region when Re_{in} is less than 2500 (or the flow rate < 3 L/min); these recirculation zones constantly moved back and forth along the outlet axial direction and changed its rotating axes from time to time. Since all the CFD simulations in this paper were steady state, they could not capture the characteristics of the unsteady flow at the outlet region well. An unsteady CFD model may be needed to better predict the flow when Re_{in} is small. When Re_{in} (or the flow rate)

increased, the flow at the outlet region became steady and recirculation disappeared. Thus, VADs optimal operation condition should consider using a higher Re_{in} . However, flow near the outlet (see the rectangular area in Figure 6 insert) is still very complicated due to the separation, and simulation results deviated more from the experimental result in this region.

As mentioned earlier, all the CFD simulations presented in this paper were performed with the MRF approach. MRF can be used when rotor-stator interaction is relatively weak. When there is a strong interaction between rotor and stator, i.e. some centrifugal pump with diffuser vane structures (28) (29), the sliding mesh approach will provide a better estimation. Since the CentriMag pump used in this paper has no diffuser vane structures and no rotor-stator interaction, MRF could provide good approximation. Taskin et. al. (25) compared the simulation results of MRF and the sliding mesh approaches in a centrifugal pump without diffuser vanes with the rotating speed of 5000 RPM and flow rate of 3 L/min. They reported a maximum 6% pressure head deviation between MRF and averaged sliding mesh simulations and fairly comparable local velocity regions. Using the sliding mesh may improve the simulation results slightly in this paper, but it dramatically increases the computational cost.

K- ω model is not included in this paper because it failed to converge. The reason is possibly because of the fast rotating motion outside the shear layer around the impeller. Furthermore, k- ϵ model is also examined, but it was not as good as the k- ϵ RNG models in predicting pressure or velocity field, and its convergence was not faster than the k- ϵ RNG model either. Since the k- ϵ RNG model was developed to improve the accuracy for rapidly strained flows and swirling flows (30), it is not a surprise that the k- ϵ model could not perform as well as the k- ϵ RNG model in this study. Thus, we excluded the k- ϵ model. Among the seven selected CFD models, k- ϵ RNG and Reynolds Stress models significantly performed better in predicting the velocity field (Figure 7), and the Laminar model showed the largest deviation from the experimental results even for the low Re_{in} which is often considered as the Laminar regime. It is notable to mention that a recent computational study of FDA benchmark nozzle by Stewart et al. (31) compared the Laminar, k- ϵ , k- ω , k- ω SST, Spalart-Allmaras models with the PIV experimental data. Their results showed that the Laminar model predicted the velocity overall the best while the k- ϵ model provided the worst results, which is opposite to the observation in this paper. Stewart et al. (31) stated that the exclusive use of the Laminar model is inappropriate but it definitely predicted the pre-transitional regions in the nozzle the best. There are many factors which may result in the opposite observation. For example, 1) the characteristics of the flow are very different between the CentriMag[®] pump in this paper and the nozzle in (31); 2) the geometry (i.e. the inlet and outlet length), meshes and the boundary conditions were different for the selected CFD models in the study of the nozzle device (31), which made it difficult to compare those CFD models side by side. However, both the study in this paper and in that of Stewart et al. (31) showed the importance of developing standards and the best approach in utilizing CFD modeling on the medical devices.

In this paper, we didn't study the effects of the parameters of individual CFD model on the simulation results. Although customized parameters may improve the accuracy of simulation, we think the procedure to find the best parameters is very difficult to standardize. The purpose of this paper is to find the best approach to numerically study the blood pump without increasing the complexity of problems. Thus, we chose the default values provided by Fluent.

Finally, CFD can be a useful tool in evaluating the medical devices. However, in order to efficiently get the most accurate results, the choice of the CFD model is important. As shown in this paper, the deviation between the results obtained with different CFD models

could be approximately 20%. With the appropriate choice of mesh, boundary conditions, and CFD model, a good velocity field can be obtained.

Conclusion

Through this study, a few widely used CFD models in developing and studying rotating circulatory assist devices were compared and validated by the experimental DPIV results. The study results demonstrated that all selected CFD models predicted some characteristics of the flow in the rotating pump well, such as the flow pattern inside the pump and the pressure. However, these CFD models provided different performances in predicting velocity field, which is important to estimate device-related blood damage. As a result, proper experimental validation may be needed for the simulated velocity field. Among the seven selected CFD models, the k- ϵ RNG models and the Reynolds Stress model predicted the velocity field the best, and the Laminar model predictions deviated the most, and the k- ω SST model, the Spalart-Allmaras model and the Transition SST model fell in the middle. It is notable to point out that the Laminar model cannot predict velocity field well when Re_{in} is small (i.e. below 2000) which is often considered in the laminar flow regime. The Reynolds Stress model didn't provide better accuracy in predicting the velocity field or pressure than the k- ϵ RNG models although it is much more computationally costly. As a result, for studies of VADs with high accuracy requirement, k- ϵ RNG models are recommended. For studies of VADs with a low accuracy requirement, Laminar model could be used due to its simplicity and low cost in computation time. It is worth mentioning that the "best CFD model" is different for the different types of flows.

Acknowledgments

This work was funded by National Institutes of Health (grant number: R01HL088100)

References

1. Murphy SL, Xu J, Kochanek KD. Deaths: preliminary data for 2010. National Vital Statistics Reports. 2012; 60(4):1–64.
2. Sondik EJ, Madans JH, Gentleman JF. Summary health statistics for U.S adults: national health interview survey, 2010. Vital and Health Statistics. 2011; 10(252):1–271. http://www.cdc.gov/nchs/data/series/sr_10/sr10_252.pdf.
3. Roger VL, Go AS, Lloyd-Jones DM, et al. Heart disease and stroke statistics-2012 update: a report from the American Heart Association. Circulation. 2012; 125(1):e2–e220. [PubMed: 22179539]
4. U.S. Department of Health and Human Services. [Accessed July 14, 2012] Organ Procurement and Transplantation Network. <http://optn.transplant.hrsa.gov/data/>. Updated July 26, 2012
5. Norman JC. Mechanical ventricular assistance: a review. Artificial Organs. 1981; 5(2):103–117. [PubMed: 7271523]
6. Behbahani M, Behr M, Hormes M, et al. A review of computational fluid dynamics analysis of blood pumps. Eur J Appl Math. 2009; 20(4):363–397.
7. Fraser KH, Taskin ME, Griffith BP, Wu ZJ. The use of computational fluid dynamics in the development of ventricular assist devices. Med Eng Phys. 2010; 33(3):263–280. [PubMed: 21075669]
8. Throckmorton AL, Wood HG, Day SW, et al. Design of a continuous flow centrifugal pediatric ventricular assist device. Int J Artif Organs. 2003; 26(11):1015–1031. [PubMed: 14708831]
9. Burgreen GW, Antaki JF, Wu ZJ, Holmes AJ. Computational fluid dynamics as a development tool for rotary blood pumps. Artif Organs. 2001; 25(5):336–340. [PubMed: 11403661]
10. Izraeliev V, Weiss WJ, Fritz B. A passively suspended Telsa pump left ventricular assist device. ASAIO J. 2009; 55(6):556–561. [PubMed: 19770799]

11. Wu J, Paden BE, Borovetz HS, Antaki JF. Computational fluid dynamics analysis of blade tip clearances on hemodynamic performance and blood damage in a centrifugal ventricular assist device. *Artif Organs*. 2010; 34(5):402–411. [PubMed: 19832736]
12. Zhu L, Zhang X, Yao Z. Shape optimization of the diffuser blade of an axial blood pump by computational fluid dynamics. *Artif Organs*. 2010; 34(3):185–192. [PubMed: 20447042]
13. Leme J, Fonseca J, Bock E. A new model of centrifugal blood pump for cardiopulmonary bypass: design improvement, performance, and hemolysis tests. *Artif Organs*. 2011; 35(5):443–447. [PubMed: 21595709]
14. Demir O, Biyikli E, Lazoglu I, Kucukaksu S. Design of a centrifugal blood pump: Heart Turcica Centrifugal. *Artif Organs*. 2011; 35(7):1525–1594.
15. Roache PJ. Quantification of uncertainty in computational fluid dynamics. *Annu Rev Fluid Mech*. 1997; 29:123–160.
16. Nigen S, El Kissi N, Piau JM, Sadun S. Velocity field for polymer melts extrusion using particle image velocimetry: stable and unstable flow regimes. *J Non-Newton Fluid*. 2003; 112:177–202.
17. Wernet MP, Subramanian A, Mu H, Kadambi JR. Comparison of particle image velocimetry and laser Doppler anemometry measurements in turbulent fluid flow. *Ann Biomed Eng*. 2000; 28(11):1393–1396. [PubMed: 11212957]
18. Huang H, Dabiri D, Gharib M. On errors of digital particle image velocimetry. *Meas Sci Technol*. 1997; 8:1427–1440.
19. Westerweel, J. PhD Thesis. Delft, The Netherlands: Delft University Press; 1993. Digital particle image velocimetry: theory and application.
20. Forliti DJ, Strykowski PJ, Debatin K. Bias and precision errors of digital particle image velocimetry. *Exp Fluids*. 2000; 28(5):436–447.
21. Anderson JB, Wood HG, Allaire PE, et al. Numerical studies of blood shear and washing in a continuous flow ventricular assist device. *ASAIO J*. 2000; 46(4):486–494. [PubMed: 10926152]
22. Curtas AR, Wood HG, Allaire PE, et al. Computational fluid dynamics modeling of impeller designs for the HeartQuest left ventricular assist device. 2002; 48(5):552–561.
23. Kido K, Hoshi H, Watanabe N, et al. Computational fluid dynamics analysis of the pediatric tiny centrifugal blood pump (TinePump). *Artif Organs*. 2006
24. Anderson JB, Wood HG, Allaire PE, et al. Computational flow study of the continuous flow ventricular assist device, prototype number 3 blood pump. *Artif Organs*. 2000; 24(5):377–385. [PubMed: 10848679]
25. Taskin ME, Fraser KH, Zhang T, et al. Computational characterization of flow and hemolytic performance of the UltraMag blood pump for circulatory support. *Artif Organs*. 2010; 34(12):1099–1113. [PubMed: 20626739]
26. Zhang J, Gellman B, Koert A, et al. Computational and experimental evaluation of the fluid dynamics and hemo-compatibility of the centrimg blood pump. *Artif Organs*. 2006; 30:168–77. [PubMed: 16480391]
27. Zhang J, Koert A, Gellman B, et al. Optimization of a miniature maglev ventricular assist device for pediatric circulatory support. *ASAIO J*. 2007; 53:23–31. [PubMed: 17237645]
28. Arndt N, Acosta AJ, Brennen CE, Caughey TK. Rotor-stator interaction in a diffuser pump. *J Turbomach*. 1989; 111(3):213–221.
29. Feng J, Berna F-K, Dohmen HJ. Numerical investigation on pressure fluctuations for different configurations of vaned diffuser pumps. *Int J Rotating Machinery*. 2007; 2007:10.1155/2007/34752
30. ANSYS. ANSYS FLUENT Theory Guide. 13. Canonsburg, PA: Ansys Fluent Inc; 2010.
31. Stewart SFC, Paterson EG, Burgreen GW, et al. Assessment of FDA performance in simulations of an idealized medical device: results of FDA's first computational interlaboratory study. *Cardiovascular Engineering and Technology*. 2012; 3(2):139–160.

Figure 1.
(a) the configuration of a CentriMag[®] blood pump in a patient (the LV apex to the aorta).
(b) the CentriMag[®] blood pump with motor.

Figure 2.
DPIV experimental setup.

Figure 3. The CentriMag[®] pump mesh for simulations. (a) side view of the vertical central plan; (b) top view of the horizontal central plane.

Figure 4.

The average error as a function of the relative average element size. X axis is the average element size relative to the medium mesh's element. Y axis is the mean error.

Figure 5. Normalized pressure difference between the inlet and outlet for various Re_{in} . Dashed lines are CFD results by difference models. Solid line with diamond symbols is experimentally measured average pressure difference.

Figure 6. Velocity magnitude at the radius of 25.3 mm with the pump rotating speed at 3000 RPM. (a) $Re_{in} = 675$ (flow rate 1L/min); (b) $Re_{in} = 2037$ (flow rate 3L/min); (c) $Re_{in} = 3393$ (flow rate 5L/min); (d) $Re_{in} = 4750$ (flow rate 7L/min). Dash lines are CFD results. Solid square symbols are the experimental DPIV results, and the error bar shows the standard errors of the experimental measurements.

Figure 7.
Global mean errors of the velocity magnitude as a function of the inlet Reynolds numbers, Re_{in} .

Figure 8.

Velocity vectors at the outlet region. Dash line with hollow arrow head is corresponding to the experimental measurement. Solid line with filled arrow head is corresponding to the laminar model results.