Available online at www.elixirpublishers.com (Elixir International Journal)

Mechanical Engineering

Elixir Mech. Engg. 46 (2012) 8211-8214

ISSN: 2229-712X

Blood flow simulation in magnetic bearing centrifugal pump by computational fluid dynamics

Mohammad Mehdi Namazi and Cyrus Aghanajafi
K.N. Toosi. University of Technology Tehran-Iran.

ARTICLE INFO

Article history:
Received: 28 February 2012;
Received in revised form:
15 April 2012;
Accepted: 5 May 2012;

Keywords
Magnetic bearing centrifugal pump,
Computational fluid dynamic,
CFX,
Blood,
Turbulence model,
Scalar stress.

ABSTRACT

Magnetic bearing centrifugal pumps are the latest model of artificial hearts for long-term use. The impeller is the only moving part in these pump which can leads to lower blood damage due to travel through the pump. In this paper computational fluid dynamics as an advanced tool is employed to simulate blood dynamics in a magnetic bearing centrifugal pump. Different simulations are performed for various ranges of device speeds (2000, 3000 and 4000 rpm) and various flow rates (1, 2,3,5,7 and 9 $\frac{L}{\min}$). As a main performance of the pump, hydraulic pressure rise is investigated. The results show that the pump can generate sufficient pressure due to different physiological activities. Also shear stress distribution specified that stress around the pump's blades (high potential regions for blood damage) is less than 98 pa which is much lower than critical stress for blood pumps.

Introduction

Every year wide range of people in all over the world require heart transplantation while, the medial community faces a serious shortage of healthy donor hearts [1]. Due to the shortage of donor hearts available, the need and demand for artificial heart pumps has been well documented. This limited number has lead to worldwide investigation and development of mechanical circulatory support such as the Ventricular Assist Device (VAD). Some of these mechanical assist devices are rotary pump. In recent years research on rotary blood pumps for totally implantable artificial hearts has been a major focus in biomedical engineering [2]. Rotary pumps in clinical use have some type of mechanical bearing that almost leads to blood damage. Therefore, they are not more reliable options for permanent implantation. Magnetic bearing centrifugal pump have been developed because they have potential for long – term use and cause little blood damage due to eliminate mechanical bearing[3]. The only moving part in these pumps is the impeller and there is no contact with fixed parts, so the regions of stagnant and high shear flow that surround a mechanical or fluid bearing are reducing. Magnetic bearing centrifugal pumps provide the potential for long life because they have no moving parts and can be designed to potentially eliminate hemolysis. Hemolysis or blood damage caused by shear stress is the break down of red cells causing the release of hemoglobin into plasma and it is very common blood trauma caused by blood pumps [4]. Blood flow dynamics behavior is involved in the mechanism of hemolysis. Analysis of the fluid dynamics in centrifugal pumps can help to evaluate the hemolysis performance of blood pumps. The blood flow in centrifugal pumps is a complex three dimensional flow. It is difficult to identify blood dynamics with experimental methods. As an advanced tool for simulation, Computational Fluid Dynamics (CFD) can predict the flow patterns in blood pumps, and can aid in the optimization of pump design. CFD is widely used to predict blood dynamics in blood pumps [5-8]. In this research, CFD package: CFX11 (ANSYS, PA, USA) was employed to simulate the flow patterns and hydraulic performance in magnetic bearing centrifugal pump.

Description and Numerical Analysis

Blood fluid properties and Pump design

Blood is a suspension of formed elements (red blood cells, white blood cells and platelets) in a Newtonian liquid (plasma). It is well known that blood is a non–Newtonian fluid that is, blood viscosity is not simple, constant ratio of shear stress to strain rate. In other word the viscosity changes with hematocrit level. Hematocrit level in heart failure patients is different from 33% to 36% [8], [9].Blood shows a Newtonian behavior for law hematocrit levels as expected for heart failure patients. Also many investigations have proved that under conditions of shear rate above 100^{-1}, the strain stressing blood proportional to the strain rate with nearly constant viscosity [11, 12].

Based on published researches, blood shows Newtonian behavior in low hematocrit levels and high shear rates which occurring blood pumps, so it is reasonable to consider blood fluid as a Newtonian with constant viscosity. The viscosity of 0.0035 $\frac{kg}{m.s}$ and a density of 1050 $\frac{kg}{m^3}$ were used for each simulation. CFD simulation provided a detail view of the flow fields in magnetic bearing centrifugal pump. So it is possible to identify the fluid mechanisms. CFD Modeling allows the characterization of flow patterns that most experimental method are not sensitive enough to discern, especially in the near – wall region. In this study Finite Volume Method (FVM) was used to carry out the CFD simulation Gambit (Fluent Inc, Lebanon, NH, and USA) was used to generate the mesh for a geometric model. The main flow path consist the inlet, the blade to blade passages, the volute and outlet. An unstructured CFD mesh was created

---

Tele:
E-mail addresses: Mehdi_namazi180@yahoo.com

for these components. Figure 1 shows the mesh containing 425k nods and 1.8 M tetrahedral cells.

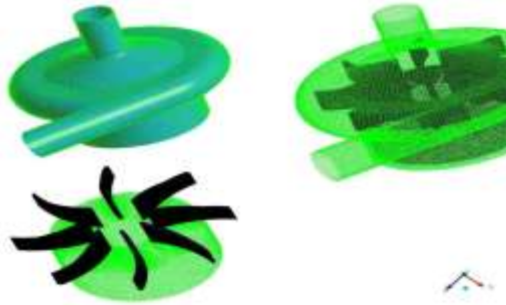

Fig.1. 3-D computational mesh for a magnetic bearing centrifugal blood pump

The full Navior-Stokes equations for flow field without heat and mass transfer is numerically solved based on FVM. By assuming the density is constant the continuity equation can be written as follows:

$$\frac{\partial \overline{U_i}}{\partial X_i} = 0 \qquad (1)$$

The momentum Conservation equation is:

$$\frac{\partial}{\partial t}\rho u_i + \frac{\partial}{\partial x_j}(\rho u_i u_j) = \frac{\partial P}{\partial x_i} - \frac{\partial \tau_{ij}}{\partial x_j} \qquad (2)$$

By using Reynolds averaging procedure for incompressible fluid, the equation 2 can be rewrite as follows:

$$\frac{\partial}{\partial t}\rho \overline{u_i} + \frac{\partial}{\partial x_j}(\rho \overline{u_i' u_j'}) = -\frac{\partial P}{\partial x_i} - \frac{\partial}{\partial x_j}(\overline{\tau_{ij}} + \rho u_i' u_j') \quad (3)$$

Where $\rho \overline{u_i' u_j'}$ representing the effect of turbulence flow field.

$$\rho \overline{u_i' u_j'} = -\rho v_t \left[\frac{\partial \overline{u_i}}{\partial x_j} + \frac{\partial \overline{u_j}}{\partial x_i}\right] + \frac{2}{3}\rho \delta_{ij} k \qquad (4)$$

$$\mu_t = \rho \upsilon_t \qquad (5)$$

The averaged viscous tensor $\overline{\tau_{ij}}$ for a Newtonian fluid is given by the following expression:

$$\overline{\tau_{ij}} = -\mu_v\left(\frac{\partial \overline{u_i}}{\partial x_j} + \frac{\partial \overline{u_j}}{\partial x_i}\right) \qquad (6)$$

Turbulence Model

The flow within pump is every where turbulent so that the effect of turbulent momentum transfers on the mean flow field.

One of more common turbulence model is the $k-\varepsilon$ Model which is two – equation Model that k is the measure of the turbulent kinetic energy contained in the flow and ε is a measure of the rate at which turbulences loses energy through viscosity dissipation.

The $k-\varepsilon$ Model calculates turbulent viscosity according to:

$$\mu_t = \rho c_\mu \frac{k^2}{\varepsilon} \qquad (7)$$

Where c_μ is a Model constant.

CFX Software and Boundary Conditions

A commercial CFD software package, CFD 11, was applied to simulate the pump performance and provide information regarding velocity profiles and pressure distribution. For each simulation mass flow rate in inlet, static pressure at the outlet and rotational speed of impeller were specified. The stationary and rotating walls were specified using Reference Frames Rotating (RFR). Flow through the pump was assumed to be steady state. Based on different flow rates which natural heart in various activities is pumping, wide range of operating conditions has been simulated flow rate from 1 $\frac{L}{\min}$ to 9 $\frac{L}{\min}$ and rotating speed from 2000 to 4000 rpm the maximum residual number

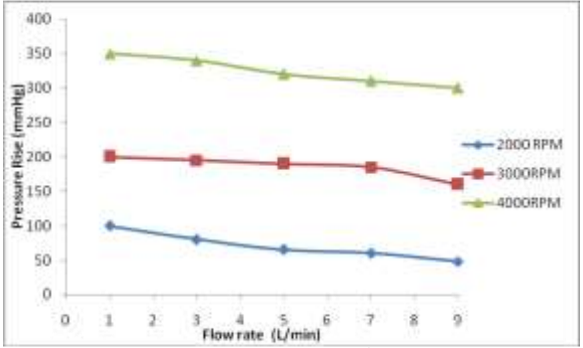

Fig.2. Pump performance curves at different rotor speeds and flow rates

Results And Discussions

By computational fluid dynamics, for the various conditions of pump operating, flow field parameters were determined. Hydraulic performance of magnetic bearing centrifugal pump, velocity profile, pressure distributions, stress and time of exposure are analyzed.

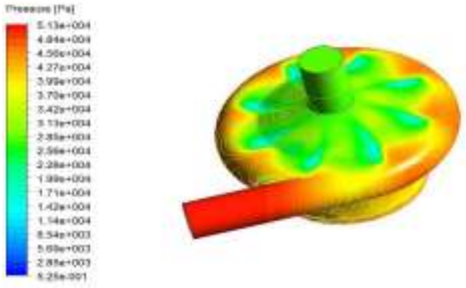

Fig.3.pressure distribution in whole model of pump

H-Q performance curve

The pump performance curve, or Head-flow curve is shown in fig 2. These data plotted with the unit of pressure as a function of flow rate ($\frac{L}{\min}$) at several rotational speeds. By increasing the rotational speed at the same flow rates, pressure difference is decreased and by increasing the rotational speeds at the same flow rates pressure differences are increased.

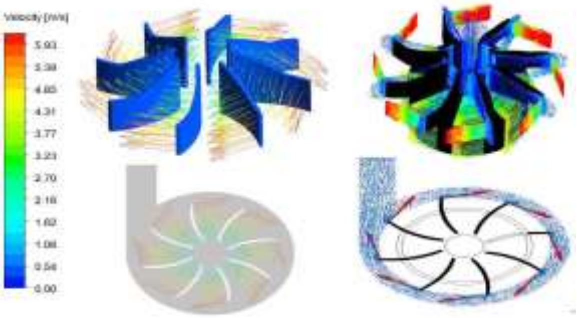

Fig.4. relative velocity vector distributions in different part of the pump

Figure 9 show that the pressure distribution in the whole pump. Based the intensity of the colors, the static pressure from inlet to outlet is continuously increased.

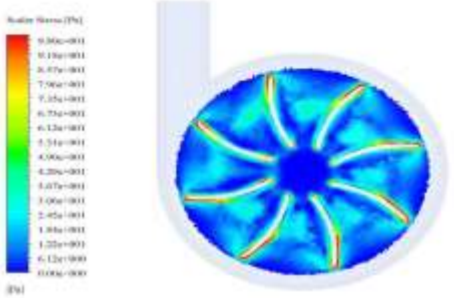

Fig.5.Scaller stress distribution around the impeller's blades

Velocity flow profile

Figure 4 presents the flow pattern through the different section of the pump. As can be seen there is no irregular flow patterns such as separation and stagnation flow regions which e could have negative effect on the blood. Also it is obvious that flow trough the volute is totally smooth.

Fluid stress distribution

Scalar stress values are directly related with the impellers rotational speed. Fluid stress distributions through the impeller under the operating condition of 5 $\frac{L}{\min}$ and 3000 rpm is shown in figure 11. Maximum scalar stress is approximately 98 pa where occur around the blade tip. Based on the published articles, the critical stress for blood pumps is 250 pa to cause blood damage. Therefore, it is predictable that blood damage in this pump is lower than others.

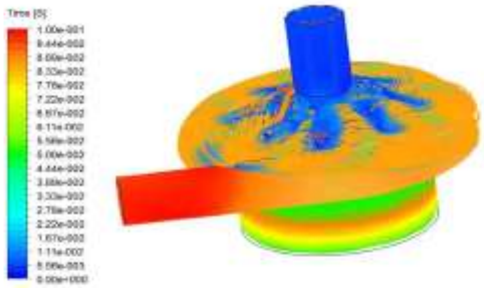

Fig.6. exposure time per second for 500 particles

Exposure time

The number of 500 particle pass through the pump from inlet to the outlet. At the pump's inlet the entire particle at the initial time equal zero for the estimating time required for each particle travel along the pump. Figure 6 illustrates the exposure time per second for 500 particles. Most of the particle travel trough the pump approximately less them 0.16 sec. only 1 percent of particles need between 0.16 to 0.36 second to exit the pump.

Conclusions

CFD is powerful analysis tools to deliver detailed insight into the complex patterns of blood flow that determine hydraulic performance. The main function of a blood pump is to deliver adequate hydraulic performance while maintaining good hematological compatibility. Shear stress distribution shows that stress around the blade tip is higher than other part of the pump but it is lower than critical stress in blood pumps. Also simulation of exposure time illustrates that only takes 0.36 second to 500 sample particles travel through the whole pump.

Reference

[1] Song, X., H. G. Wood, et al., 2004, Computational Fluid Dynamics (CFD) study of the 4th generation prototype of a continuous flow Ventricular Assist Device (VAD). *Journal of Biomechanical Engineering* 126(2): 180-187.

[2] Timms, D. L., A. C. C. Tan, et al., 2005, LVAD Pump Performance and Force Characteristics In A Pulsatile Complete Mock Circulation Loop. *Artificial Organs* 29(7): 573-580.

[3] Tsukamoto, Y., K. Ito, et al., 2001, Computational Fluid Dynamics analysis for centrifugal blood pumps. *Journal of Congestive Heart Failure and Circulatory Support* 1(4): 337-343.

[4] Nishida, M., Asztalos, B. and Yamane, T., 1999, Flow visualization study to improve hemo- compatibility of a Centrifugal Blood Pump. Artificial Organs, 23, 697 – 703.

[5] Verdonck. P., 2002, The role of computational fluid dynamics for artificial organ design. Artificial Organs, 26, 569 – 570.

[6] Song, X.W., Throckmorton, A.L., Wood, H.G., Antaki, J.F. and Olsen, D.B., 2003, Computational fluid dynamics predication of blood damage in a centrifugal pump. Artificial Organs, 27, 938 – 941.

[7] Sukumar, R., Athavale, M.M., Makhijani, B.V., 1996, Application of computational fluid dynamics techniques to blood pumps. Artificial Organs, 20, 529 – 533.

[8] Leverett, L. B., Hellums, J. D., Alfrey, C. P., et al. 1972, ''Red blood cell damage by shear stress.'' Biophys. J., 12, 257–73.

[9] Jahanmir S, Willis JP, Tomaszewski MJ, Hryniewicz P, Heshmat H., 2004,Status of magnetically suspended MiTi Heart LVAD. ASAIO J; 50:111.

[10] Behr, M., Arora, D., Nos´ e, Y. & Motomura, T., 2004, Performance analysis of ventricular assist devices using finite element flow simulation. Int. J. Numer. Meth. Fluids 46,1201–1210.

[11] Giersiepen M, Wurzinger LJ, Opitz R, Reul H. ., 1990, Estimation of shear stress-related blood damage in heart valve prostheses in vitro comparison of 25 aortic valves. Int J Artif Organs; 13: 300–6.

[12] Mussivand, T., 2008, Mechanical circulatory support devices: is it time to focus on the Complications, instead of building another new pump? Artif Organs; 32(1): p. 1-4.R.

[13] Simon, M.A., et al., 2008, Current and future considerations in the use of mechanical Circulatory support devices. Annu Rev Biomed Eng; 10: p. 59-84.

[14] Bhama, J.K., et al., 2009, Clinical experience using the Levitronix CentriMag system for temporary right ventricular mechanical circulatory support. J Heart Lung Transplant; 28(9): p. 971-6.

Nomenclature

P	Pressure (Pa)
Q	Flow rate (l/min)
x_i	Components of all forces
U_j	Velocity vector components
H	Head rise (mm Hg)
τ_{ij}	Viscous stress tensor
υ_t	Turbulent diffusivity
K	Turbulent kinetic energy
μ_υ	Turbulent viscosity
ρ	Density (kg/m^3)
μ	Dynamic viscosity (kg/ms)