Proceedings of the 29th Annual International
Conference of the IEEE EMBS
Cité Internationale, Lyon, France
August 23-26, 2007.

Flow Simulation of a Diaphragm-type Ventricular Assist Device with Structural Interactions

M. H. Moosavi and N. Fatouraee

Abstract— **A numerical simulation of the fluid and structure interactions was performed for a diaphragm-type Ventricular Assist Device (VAD). The simulation was included two incompressible fluids and three elastic solid regions. The detailed information of both fluid and solid dynamic are crucial to evaluate the performance of the device. Localization of potential points prone to hemolysis and thrombus formation and disturbed flow with creation of recirculation zones are vary important. Here, the HeartSaver VAD was modeled as a two dimensional chambers with an inlet and an outlet elastic valves, an elastic sac-shape membrane, driving fluid and blood. A pulsatile velocity condition was applied at the inlet of the driving fluid chamber. As the results, the flow characteristics affecting the blood cells including velocity, pressure, wall shear stress and the elastic valves operation were presented in details.**

I. Introduction

In spite of the recent noticeable achievements in science and technology, heart diseases are the leading factor of death. One of the most severe types of heart disease is heart failure, in which the heart cannot pump a sufficient amount of blood through the body. At present, its most conventional treatment method is provided by using the heart transplant, but there is little access to donor hearts and in some conditions the patient heart needs just a little help for its proper operation [1]. Some mechanical devices are developed for overcoming these problems. Ventricular Assist Devices (VADs) are developed to help the heart to pump the required outlet flow when the heart is failed to do the proper operation. According to the situation of failure in the heart, these devices usually mount between left ventricular and aorta, or between right ventricular and pulmonary vessel. The available VADs have wide applications for patients whom heart need assist before and after surgery and the patients who wait for a total heart replacement. These increasing considerations on VAD tools made it a proper device on current engineering attempts.

Some biological attentions must be paid in design and manufacturing of these devices. In order to reduce the red blood cell damage, the pumping structure must provide the conditions to prevent high shear stress and flow separation regions. Additionally, the blood must be washed out properly since the formation of thrombus tends to occur within stagnation regions [2]. Since the operating conditions are severe, it is very difficult to use instrumentation for flow measurements. Therefore it became necessary to look at the flow by using computational techniques. Computational analysis can also be used to optimize the design of mechanical devices at a significantly lower cost and time than required by an empirical approach.

There are many different types of VADs. The present study however is concerned with the diaphragm-type VAD. For numerical analysis of non diaphragm-type VADs, different reports have been published [3,4], but because of its complex flow conditions the numerical studies for diaphragm-type VADs are limited. In these devices the blood pumping action is affected by transferring driving fluid between a volume displacement chamber and the pumping chamber of the VAD through one-way valves. Therefore the interactions of both driving fluid and blood with the elastic diaphragms make the solution of flow domain with more difficulties. Hence the current studies on these devices are accompanied with many simplifications. Ottawa University's HeartSaver system is one of the first diaphragm-type assisting devices which are considered for numerical studies [5]. Firouzi et al. [6] have modeled the two dimensional flow domain of this device with assuming a rigid predefined motion on the diaphragm. Doyle [7] developed his modeling on idealized Brunel University VAD by concerning both the driving fluid and blood dynamics. This study provided more realistic modeling of flow by solving a Fluid-Structure interaction domain but in this modeling there is an assumption on ignoring the valves motion. The inlet and outlet valves were all closed in systole and diastole phases, respectively. Shim et al. [8] used a fully coupled method to compute the flow interacting with a flexible sac-VAD chamber, they used on/off boundary conditions to model the valves action. Their model ignored not only the opening and closing phases of the valves, but also the effect of the valves on profile and direction of the entering flow. Avrahami et al. [9] were the only ones to use a realistic 3D model with valve geometry to simulate a sac-type VAD. Their model ignored only the motion of valves leaflets.

Here we developed a more complete model of the diaphragm-type assist devices by considering Fluid-Structure interaction. A two dimensional model of

Manuscript received June 25, 2007.

M. H. Moosavi is a PhD candidate in the Biomedical Engineering Faculty, Amirkabir University of Technology (Tehran Polytechnic), Tehran, Iran (e-mail: moosavi_mh@bme.aut.ac.ir).

N. Fatouraee is an assistant professor in the Biomedical Engineering Faculty, Amirkabir University of Technology (Tehran Polytechnic), Tehran, Iran (Corresponding author, phone: +98-21-64542368; fax: +98-21-66468186; e-mail: Nasser@ aut.ac.ir).

HeartSaver VAD system is proposed for this modeling. HeartSaver belongs to pulsatile flow pump family. As illustrated in figure 1, it consist of two chamber separated by a flexible diaphragm. One of the fluid chambers contains blood and inlet and outlet valves. The other chamber contains a driving fluid, which is oscillating periodically into and out of the chamber through a pumping action. As the diaphragm moves downwards with the influence of driving fluid, low blood pressure is generated below the inlet valve. The inlet valve is pushed down to open, and the blood is sucked into the pump. As the diaphragm moves upward with the influence of driving fluid, high blood pressure is generated below the inlet and outlet valves. The high blood pressure pushes the inlet valve back to the closed position, and then pushes the outlet valve up to open. This allows the blood through the outlet valve to go out of the pump. As the diaphragm moves downwards again, the previous cycle is repeating. The analysis includes the effect of the diaphragm movement and its deformation, the FSI between the diaphragm and both blood and driving fluid, the FSI between the inlet/outlet valves and the blood, and the solid-solid contact between the inlet/outlet valves and the valve seats.

II. Method

A. Geometry of model

Figure 1 illustrates the geometry of the model of HeartSaver assist device which is used in this study. A two dimensional model of this device is used in this modeling. Although the real geometry consists of more complex three dimensional elements, we simplified it to reduce the computational time and check the feasibility features and find the associated problems with this type of modeling. Indeed special cares are given to the inlet and outlet conditions and they scaled to the two dimensional geometry. The size of all parts in this model is obtained from the real dimensions [10].

B. Governing Equations

Blood and the driving fluid are modeled as laminar, isothermal and Newtonian viscous fluids. Because of pulsatile flow in this diaphragm-type VAD, the transient form of the governing equations considered in this formulation. Both blood and driving fluids are modeled as

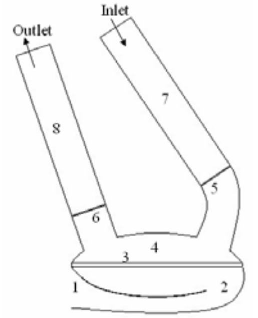

Fig. 1. Schematics of HeartSaver VAD which is used in this modeling.

slightly compressible a fluid which ignores the change of density due to pressure in the momentum equation, but not in the continuum equation (it make the convergence of the solution more difficult). Such an assumption is necessary because incompressible fluids can not compress in an enclosed container. Therefore the momentum and continuity equations for this type of modeling are:

$$\frac{1}{\beta}\frac{\partial p}{\partial t}+\frac{\partial u_i}{\partial x_i}=0 \tag{1}$$

$$\rho\frac{\partial u_i}{\partial t}+\rho\left(u_j-\frac{\partial d_j^f}{\partial t}\right)\frac{\partial u_i}{\partial x_j}=\frac{\partial \tau_{ij}^f}{\partial x_j} \tag{2}$$

where β is the bulk modulus, p is the pressure, u_i is the flow velocity along axis i and d_j^f is the flow displacement along the fluid-structure interface or other moving boundary. The stress tensor can also be presented as below:

$$\tau_{ij}^f=-p\delta_{ij}+2\mu e_{ij} \tag{3}$$

δ_{ij} is the Kronecker delta and μ is the viscosity. The strain tensor e_{ij} is defining as below;

$$e_{ij}=\frac{1}{2}\left(\frac{\partial u_i}{\partial x_j}+\frac{\partial u_j}{\partial x_i}\right) \tag{4}$$

The solid is mathematically modeled using the classical Lagrangian formulation, which is as follows;

$$\frac{\partial \tau_{ij}^s}{\partial x_j}=\rho^s\frac{\partial^2 d_i^s}{\partial t^2} \tag{5}$$

where τ_{ij}^s is the Cauchy stress tensor, d_i^s is the solid displacement component and ρ^s is material density. In this formulation the coupling condition of fluid and structure interaction must be satisfied. The kinetic condition which denotes the no-slip conditions at the interface are:

$$d_i^f=d_i^s \tag{6a}$$

$$\frac{\partial d_i^f}{\partial t}=\frac{\partial d_i^s}{\partial t} \tag{6b}$$

The kinetic coupling condition denotes the equilibrium of forces as below:

$$n_j\tau_{ij}^f=n_j\tau_{ij}^f \tag{7}$$

where n is the unit vector normal to the interface.

C. Boundary Conditions

The boundary conditions for the solid regions are: (1) both sides of the diaphragm and one side of both valves are fixed on the solid model, (2) the seat and valve surfaces are defined as a contact pair, (3) FSI conditions is defined between diaphragm and both blood and driving fluids, also between valves and blood.

The boundary conditions for the fluid regions are: (1) a no-slip conditions beside the walls, (2) as for the solid model, FSI conditions between diaphragm and both blood and driving fluids, also between valves and blood, (3) Gap condition for lines connecting the free ends of both valves to the seat. The gap condition makes an interval close or open according to the length of the gap line. If the valve is very close to the seat, the line become shorter than the predefined parameter, then the boundary condition of this line will switch to wall-type and prevent the flow going through the gap. If the valve is away from its seat by a certain distance, as measured by the length of the gap line, the wall condition will turn off. The gap line now becomes a normal interior line and the flow can go through it. (4) Oscillatory velocity condition for the inlet of driving fluid as the relation $u=u_0 \cos(2\pi t/T)$. T is the period of heart beat which is set equal to 0.87 sec [6]. For calculation of u_0 special cares are given to the inlet and they scaled with the two dimensional geometry. We assume a 0.005 m^3/min volumetric blood flow rate and a 10 cm width of the three dimensional HeartSaver system. Therefore a 0.050 m^2/min flow rate must consider in this 1 m width 2D model. u_0 will be calculated by taking the fluid flux from the inlet equal to the obtained flow rate which is equal to 0.13 m/sec. Our experiments with the obtained velocity profile provide proper motion of diaphragm.

D. Physical Property of Material

Two fluids are used in this modeling, water for the external chamber and blood for the internal HeartSaver chamber. The following material properties are necessary for fluid materials: density, viscosity, bulk modulus of elasticity (for slightly compressible fluid only). The two flexible valves, their seats and the diaphragm have been modeled as linear elastic materials. The necessary parameters for these models including: the Young's modulus, the Poisson ratio, and the solid density. Table I and II present these parameters for the fluid and solid regions, respectively.

E. Simulation Process

With the use of the Lagrangian formulation in the structural model and the Arbitrary Lagrangian-Eulerian (ALE) formulation in the fluid counterpart, the fully coupled system was solved using ADINA (ADINA™, version 8.2, Automatic Dynamic Incremental Nonlinear Analysis, Watertown, MA). An iterative Newtonian method is used for solving the obtained system of equations. For the solid equation and the fluid equation the number of iterations and iteration tolerance is set to 100 and 0.001, respectively.

F. Computational Grid Generation

Fluid domain is meshed with 37299 2-D Fluid Planar type elements. All elements are triangular and contain three nodes. The elements are refined where a complex flow domain expected as shown in Fig. 2. For the solid region 1893 2-D Solid Plane strain type elements are used. These rectangular elements contain 9 nodes. The mesh-independency of the simulation results is achieved by considering deference mesh configurations.

III. Results

The initial and operating conditions in this modeling are described below. Initial condition involves the diaphragm to be full of driving fluid and the inlet and outlet valves closed. At t=0, the diaphragm starts to move due to the inlet conditions. Conventionally three cycles is sufficient to obtain steady repeated pulsating flow domain [6]. Our experiments also show the results of third and fourth cycles similar and then we used the third cycle of oscillation for evaluation. Figures 3 and 4 present the velocity fields of blood and driving fluid in systolic and diastolic phases respectively. They also show the blood flow due to the movement of diaphragm in and out of the pump. The opening and closing of the inlet and outlet valves and the related flow field can be clearly distinguished from these figures. Both valves have proper behavior because of the large adverse pressure gradient prior to the valve closing. The recirculation flows are appeared in the blood chamber and after the elastic valves. They are generated due to separation of flow near the structures. Review of the blood flow domain at different time steps show this recirculation zone moving from the inlet to outlet by exchanging the phases from systole to diastole. The wakes appeared after the outlet valve is continuously moving toward the outlet. The appearing of these wakes prevents the blood to be coagulated. Only at the end of diastole phase, some stagnation points are appeared near the inlet valve. According to these results, the maximum blood flow parameters such as velocity, velocity gradient and shear stress are occurring near the valves when they are open.

Pressure distribution at the beginning of systole phase, when the diaphragm is going to push the blood out of the chamber, decreases toward the outlet of chamber. Similarly its distribution is increasing from the inlet to the diaphragm during the diastole phase.

TABLE I
MATERIAL PROPERTIES OF FLUIDS

	Density K/m^3	Bulk Modulus N/m^2	Viscosity Kg/m.s
Water	1000	10^{13}	0.001
Blood	1105	3×10^8	0.00466

TABLE II
MATERIAL PROPERTIES OF SOLIDS

	Density Kg/m^3	Young Modulus N/m^2	Poisson Ratio
Valves	1000	6.5×10^9	0.495
Seats	1000	200×10^9	0.495
Diaphragm	1000	10^8	0.495

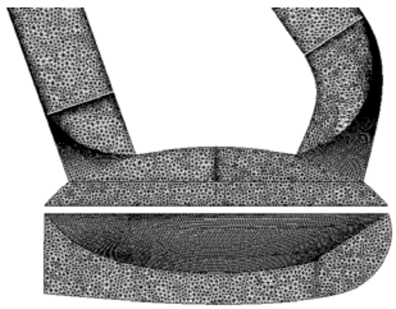

Fig. 2. Computational mesh at fluid regions of the proposed model.

Figure 5 also shows a comparison of flow rate at the driving fluid port, blood inlet and outlet valves at different time steps. The driving flow rate similar to its velocity condition provides a cosine wave form. Initially both the driving fluid and blood outlet provide positive flow rate, but at the inlet its value is zero. As the driving flow changes the actuation to suction mode, the blood outlet tends to be closed and the inlet gradually opens. The horizontal line in this figure indicates the time periods that the valves are closed. The drop from horizontal line after a peak is due to the back-flow immediately before closing the valve. Both the inlet and outlet valves show this backflow because of the adverse pressure gradient prior the valve closing. The backflow from these valves usually identified as a major factor that causes the loss of pumping efficiency and can be measured from these computations. This study has considered both the driving fluid and blood to be laminar. Our results indicate that the peak filling Reynolds number was less than 1900. In such flow conditions, the effect of transitional flow in the inlet cannula during deceleration phase of filling is considered to be negligible [11]. However, according to Konig et al. [12], sudden expansion of the flow as it enters the main chamber might tend to cause transient turbulence even at Reynolds numbers as low as 750. Therefore, assumption of laminar flow may introduce some inaccuracies, especially downstream from the valves.

Red blood cells can be subjected to shear stress up to 150 N/m^2 before significantly release of hemoglobin content occurs [13]. Results show that the magnitude of maximum shear stress at each time step occurred at the end of leaflet and their total maximum is less than 50 N/m^2. Therefore the model of leaflets doesn’t perform a critical shear stress on red blood cells.

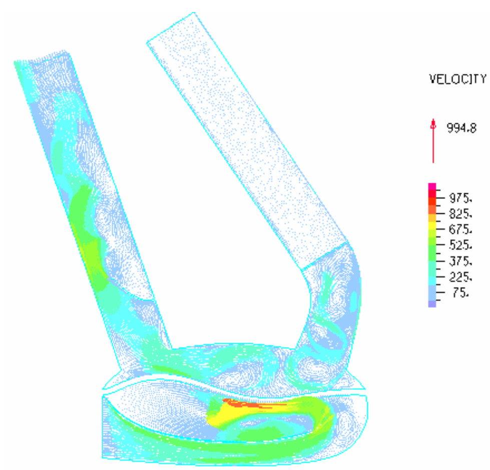

Fig. 3. Flow domain during systole phase.

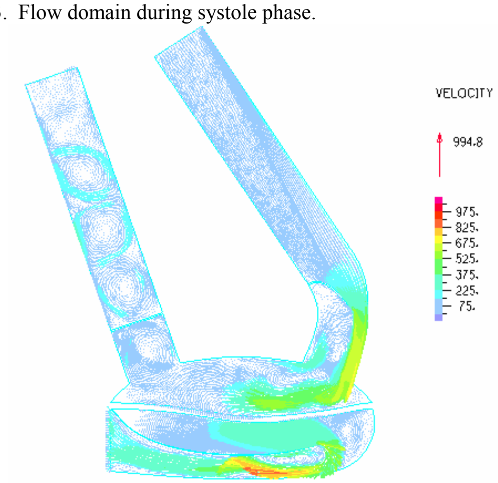

Fig. 4. Flow domain during diastolic phase.

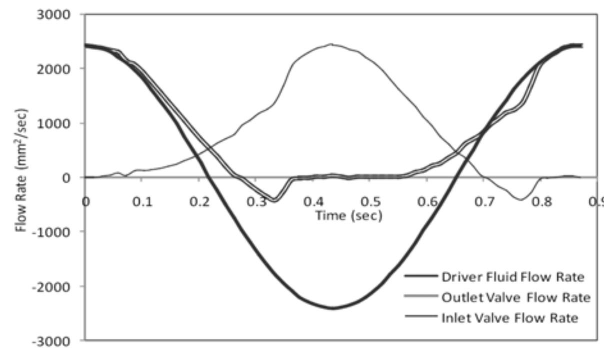

Fig. 5. Flow rate at driving fluid port and blood inlet and outlet.

References

[1] E. Collard, M. J. Van Dyck, and L. M. Jacquet, “Ventricular assist devices,” *Current Opinion in Anaesthesiology*, vol. 16, pp. 33-43, 2003.

[2] C. Kiris, D. Kwak, and R. Benkowski “Incompressible Navier-Stokes Calculations for the Development of a Ventricular Assist Device,” *Computer & Fluids*, vol. 27, pp. 709-719, 1998.

[3] G. W. Burgreen, and J. F. Antaki, “Computational Fluid Dynamics as a Development Tool for Rotary Blood Pump,” *Artificial Organs*, vol. 25, no. 5, pp. 336-340, 2001.

[4] M. Pinotti, and E. S. Rosa, “Computational Prediction of Hemolysis in a Centrifugal Ventricular Assist Device,” *Artificial Organs*, vol. 19, no. 3, pp. 267-273, 1995.

[5] T. Mussivand, K. D. Day, and B. C. Naber, ”Fluid Dynamic Optimization of a Ventricular Assist Device Using Particle Image Velocimetry,” *ASAIO J*, vol. 45, no. 1, pp. 25-31, 1999.

[6] F. Firouzi, N. Fatouraee, and S. Najarian, “Simulation of Blood Flow in the Sac-Type Ventricular Assist Device Using Computational Fluid Dynamic,” *Iranian Journal of Biomedical* Engineering, vol. 1, no. 2, pp. 129-142, 2005.

[7] M. G. Doyle, “Simulation of Blood Flow in Ventricular Assist Device with Fluid-Structure Interaction effects,” M.S. thesis, Ottawa Univ., Ottawa, Canada, 2003.

[8] E. B. Shim, J. Y. Yeo, H. J. Ko, et al., “Numerical analysis of the three-dimensional blood flow in the Korean artificial heart,” Artificial Organs, vol. 27,no. 1, pp. 49-60, 2003.

[9] I. Avrahami, M. Rosenfeld, S. Raz, S. Einav, “Numerical Model of Flow in a Sac-Type Ventricular Assist Device” Artificial Organs, vol. 30,no. 7, pp. 529-538, 2006.

[10] T. Mussivand, “Electrohydraulic Ventricular Assist Device,” United State Patent, No. 5569156, 1996.

[11] L. Ge, S. Jones, F. Sotiropoulos, T. Healy, A. P. Yoganathan, “Numerical simulation of flow in mechanical heart valves: grid resolution and the assumption of flow symmetry,” J Biomech Eng., vol. 125,no. 5, pp. 709-718, 2003.

[12] C. S. Konig, C. Clark, M. R. Mokhtarzadeh-Dehghan, “Investigation of unsteady flow in a model of a ventricular assist device by numerical modeling and comparison with experiment,” *Medical Engineering and Physics,* vol. 21,no. 1, pp. 53-64, 1999.

[13] Y. C. Fung, *Biomechanics: Mechanical Properties of Living Tissue*, (2nd ed.), Springer, New York, 1993.