Publishing House
AKAPIT

COMPUTER METHODS IN MATERIALS SCIENCE
Informatyka w Technologii Materiałów

Vol. 11, 2011, No. 1

NUMERICAL ANALYSIS OF AN INFLUENCE OF THE ARTIFICIAL VALVE TYPE ON THE BLOOD FLOW INSIDE THE VENTRICULAR ASSIST DEVICE

PRZEMYSLAW KLOSINSKI, PIOTR REOROWICZ, DAMIAN OBIDOWSKI*, KRZYSZTOF JOZWIK

Technical University of Lodz, Institute of Turbomachinery,
Medical Apparatus Division, Lodz, Poland
**Corresponding author: damian.obidowski@p.lodz.pl*

Abstract

A Ventricular Assist Device (VAD) is used in case of severe heart illnesses when the natural heart supplies the body with an insufficient volume of blood. Any damage or improper functioning of the VAD can result in the patient's death. This implies the constant need to improve the design of VADs and artificial valves which are crucial parts of the device.

The authors used the latest Computer Aided Design and Computational Fluid Dynamics software to analyze the flow in the pneumatic Ventricular Assist Device designed at the Foundation of Cardiac Surgery Development and equipped with two different types of valves. In the study, a single-disc mechanical artificial heart valve based on the invention of Prof. J.J. Moll, modified at the Institute of Turbomachinery, TU Lodz, was compared with a three-leaflet polyurethane artificial heart. A comparison was made on the basis of the flow visualization inside the VAD chamber and the size of stagnation regions where the flowing blood may coagulate. An angular position of the disc valve was determined on the basis of the previous studies. A steady state simulation was performed on the assumption that walls of the assist device, adapters and valves were rigid. Dynamic viscosity of blood was defined on the basis on the Non-Newtonian Power Law. Simulations were preformed for systole and diastole conditions. The Ansys CFX v12 code was used to perform preprocessing, solving and postprocessing stages. Deformations of the three-leaflet polyurethane valve were obtained in SolidWorks 2009 and imported to Ansys ICEM v.12.

On the basis of the preformed analysis, it has been proven that the disc mechanical heart valve generates better flow conditions inside the heart chamber, especially if a risk of coagulation is concerned. Moreover, the flow observed inside the chamber when the disc valve was used is more homogenous and a single swirl occurring in the central part enables good washing of the connection of the diaphragm and chamber regions.

Key words: ventricular assist device, artificial heart, artificial heart valves, numerical fluid mechanics

1. INTRODUCTION

Heart diseases are one of the most frequent causes of death and they are serious threats to human life. A proper diagnosis procedure may help to treat heart diseases. In many cases the only available procedure used to rescue the patient's life is heart transplantation. An insufficient number of donors is a significant problem not only in the heart illness treatment, but since heart malfunction is always dangerous to life, many efforts are made to create an artificial heart that may be implanted in place of the human heart. This will make the treatment independent of a limited number of donors. The usage of an external artificial heart that supports the patient's heart by forcing an extra blood flow rate is one of methods that may help to heal heart or to extend the time the recipient may wait for transplantation. The presented research is devoted to examination of an effect of the artificial heart valve type on the flow

 ISSN 1641-8581

inside the chamber and in the adaptors. The numerical experiment was conducted with the non-modified Ventricular Assist Device developed and provided by the Foundation of Cardiac Surgery Development (Bujok et al., 2010). Two different types of artificial valves were introduced into the VAD. A single-disc mechanical valve based on the original design of Prof. J.J. Moll, modified in TU Lodz, was the first type, the second one was a three-leaflet polyurethane valve developed at the Foundation of Cardiac Surgery Development.

An evaluation of different solutions was made on the basis of the flow visualization inside the VAD chamber and the spread of blood flow stagnation regions.

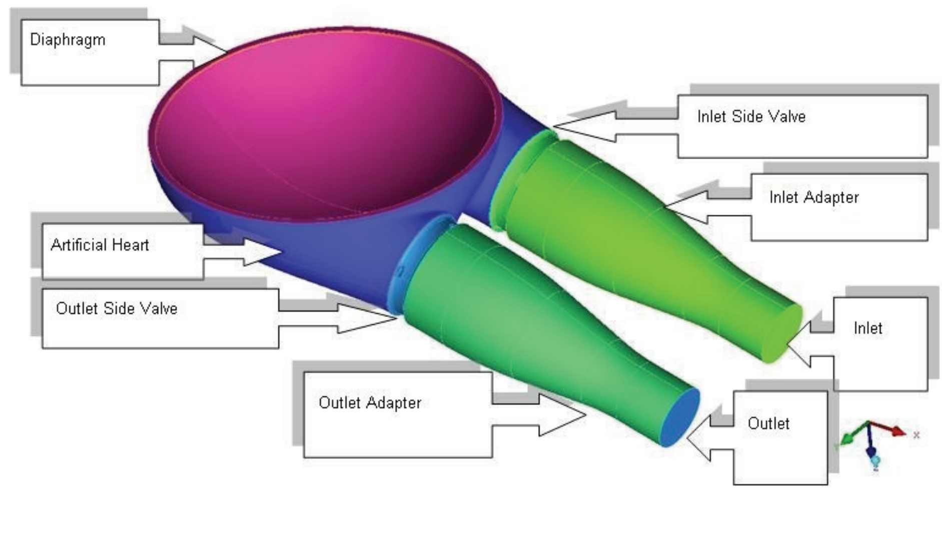


Fig. 1. Computational domain with the described model elements.

The analysis was made using the most advanced commercial software tools. The VAD and three-leaflet valves were designed with the SolidEdge code. The single disc valve was designed in the Pro-Enginner code, then models were integrated in the ICEM software, in which the discretization of domains was also performed. Ansys CFX v12 was used to conduct the numerical experiment. The non-Newtonian model of blood, based on the Power Law approach, was applied. Two stages of the device operation were simulated. Diastole – for which the diaphragm is in its most external position and the volume of blood inside the chamber is the highest. In this particular time of the heart operation cycle, the inlet valve is totally open and the outlet valve is almost closed (5% open). The opposite deflection of the diaphragm – systole – occurs if most of blood is discharged from the chamber. The outlet valve is totally open, whereas the inlet valve is almost closed (5% open). Finally, diastole of the VAD equipped with three-leaflet valves was compared with diastole of the VAD with single disc valves. The same was done for systole conditions.

All computations were performed at the Institute of Turbomachinery, Technical University of Lodz.

2. METHOD

A model of the pneumatically driven Ventricular Assist Device – POLVADEXT – was used in the presented study. The whole model consists of a blood chamber, an air chamber separated by a diaphragm, two artificial heart valves, and two adapters (figure 1). The pneumatic part of the VAD is irrelevant in this study, hence it is omitted.

The CFD software solves the Navier-Stokes equations, employing the Finite Volume Method, thus the computational domain has to be divided into small volumes. The geometry was imported into Ansys ICEM v12, in which it was discretized. Due to the complexity of the geometry, an unstructured mesh was used. In the boundary layer, in the vicinity of walls, prismatic elements were employed to solve accurately the flow in regions of the highest velocity gradients. In all the presented cases, a number of elements in the whole domain exceeded 8 millions.

The Shear Stress Transport turbulence model was used. The quality of the mesh was checked in a mesh independence study and with the Yplus parameter that was smaller than 8 in all cases. Low residual levels were achieved in the solution. The contour plot presented in Figure 2 shows a distribution of Yplus on the surface of the 5% open single disc valve. One can notice that the highest values of this parameter are observed in the gap between the disc and the ring of the valve. This is due to the highest increase in the velocity as the channel narrows significantly. The maximum value of Yplus parameter in this region is displayed in the legend and equals 5.33, which proves a high quality of the mesh.

Boundary conditions were estimated on the basis of the literature survey. At the inlet, a velocity profile was introduced with the maximum value listed in table 1. The profile was based on the cork shape, typical of turbulent flows. It is described by the following formula:

$$V = V_{\max}\left|\left(1-\frac{r}{R}\right)^{\frac{1}{7}}\right| \tag{1}$$

where: V – velocity at the particular node of the mesh at a distance r measured from the axis of the adapter, V_{max} – velocity along the axis of the adapter, R – radius of the adapter, r – distance from the axis of the adapter.

At the outlet cross-section, pressure was assigned. Velocity and pressure values used in the presented study are listed in table 1.

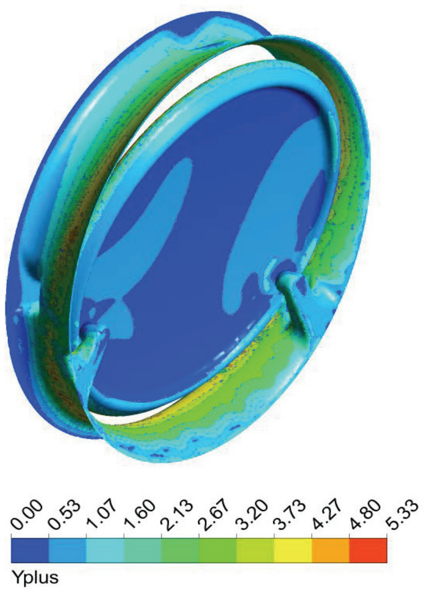


Fig. 2. Yplus parameter contour plot on the surface of the single disc valve.

Table 1. Boundary conditions for all cases under investigations

	Maximum inlet velocity, m/s	Outlet static pressure, kPa
Diastole	3.815	13.65
Systole	0.315	-0.2821

The non-Newtonian blood model based on the Power Law (Walburn et al., 1976, Gijsen et al., 1999, Johnston et al., 2004, Obidowski et al., 2008, Jozwik & Obidowski 2010) model was used in the study. The blood dynamic viscosity was described as a function of strain rate. The Basic Power Law model was limited in the range of very low strain rate values (below 1×10^{-9} s^{-1}). It is known that the Power Law in the range of high strain values underestimates values of dynamic viscosity. Thus, for strain rates higher than 327 s^{-1}, the Newtonian model is proposed, for which the value of dynamic viscosity is constant and equal to 0.00345 Pa.

The dynamic viscosity used in the study is described by the following equation:

$$\begin{cases} \mu = 0.554712 \quad \text{for} \quad \dfrac{\partial v}{\partial y} < 1\times10^{-9} \\ \mu = \mu_0\left(\dfrac{\partial v}{\partial y}\right)^{n-1} \quad \text{for} \quad 1\times10^{-9} \le \dfrac{\partial v}{\partial y} < 327 \\ \mu = 0.00345 \quad \text{for} \quad \dfrac{\partial v}{\partial y} \ge 327 \end{cases} \tag{2}$$

where: μ – dynamic viscosity, $\dfrac{\partial v}{\partial y}$ – strain rate.

3. RESULTS

The most dangerous complication related to an application of the VAD is blood coagulation followed by thrombus formation. It may occur if blood platelets are activated by deformation or contact with air or materials of the implanted device. Another undesired phenomenon occurring during the VAD operation is haemolysis mostly promoted by strong turbulent regions at which platelets may be activated.

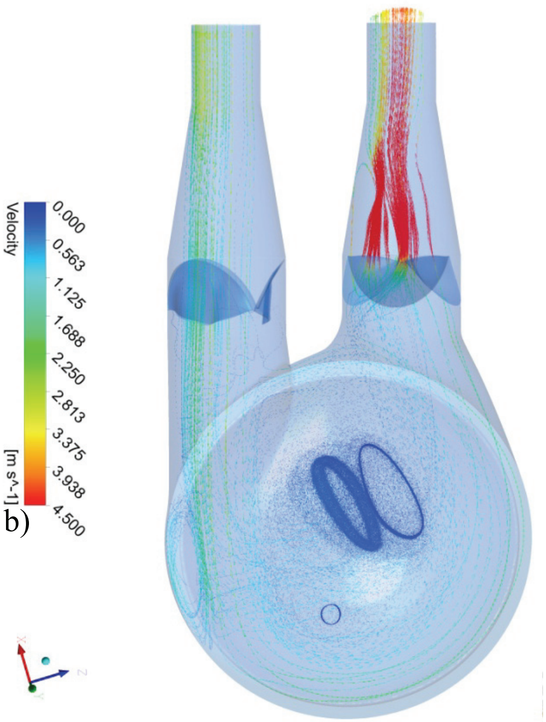
A risk of coagulation increases if the exposure time is elongated. This leads to a conclusion that regions of very low velocity are potentially the most dangerous, especially in the vicinity of VAD walls. At the same time strong turbulences and high shear stresses may induce platelet activation, thus high velocity gradients should be avoided as well.

Two types of visualization methods are used in the presented paper, namely: streamlines with velocity vectors (figure 3) and surfaces enclosing low velocity regions (figures 4, 5).

Flow characteristics during the diastole stage of the heart operation cycle are compared in the VAD with different types of the valves in figure 3. It can be easily noticed that the flow in the blood chamber with single disc valves is more regular. Higher velocities (greenish lines) are also visualized in the connection of the diaphragm and the chamber. This ensures lower risk of coagulation due to a short contact time of blood platelets with the material of the chamber than in the case of the VAD with three-leaflet valves.

It has been shown in the earlier study (Jozwik et al., 2009, Jozwik et al., 2009,) that the angular positioning of the disc valve plays a significant role in the proper stream optimization. In this paper, the best angular position is presented for all the cases. This has no influence on the flow if three-leaflet valves are considered.

a)



b)

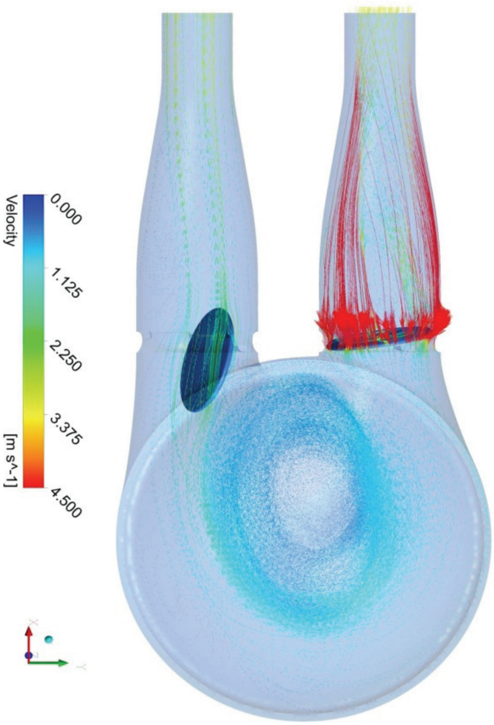
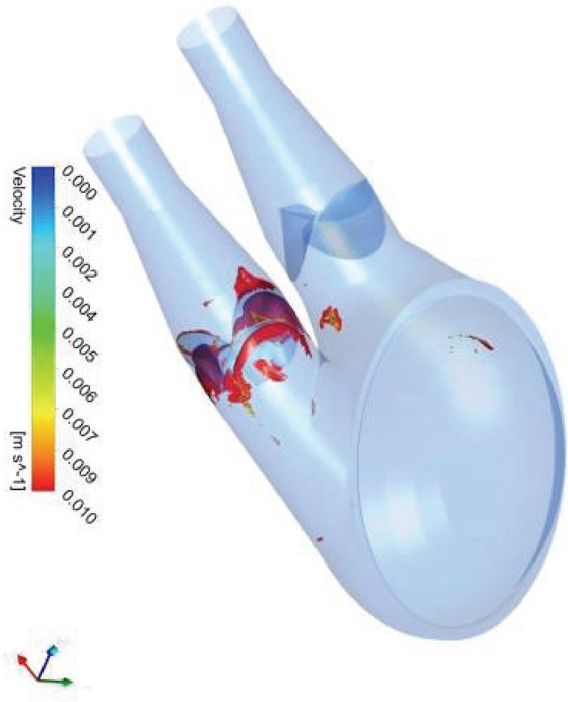


Fig. 3. Streamlines with velocity vectors during diastole, a) three-leaflet valve b) single disc valve.

a)



b)

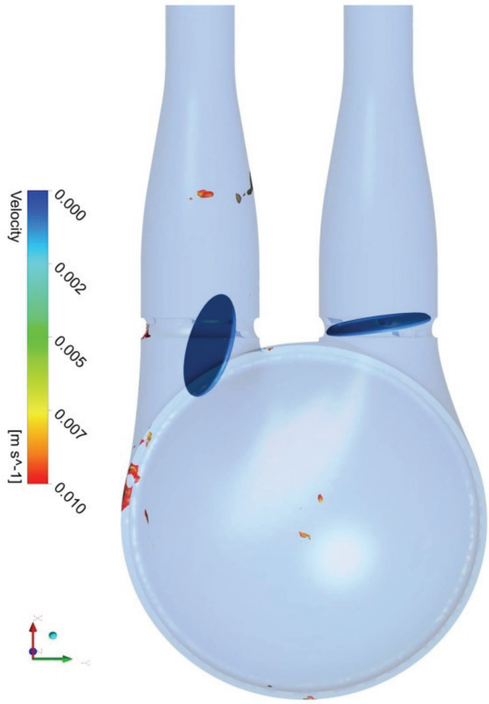


Fig. 4. Stagnation regions during diastole, a) three-leaflet valve b) single disc valve.

The polyurethane valve (see figure 3a)) causes highly worse flow conditions than the single-disc valve (see figure 3 b)). The streamline vector plots shown in figure 3a present several regions of swirls within the chamber that do not lie along one axis. Moreover, a separation is observed near the wall just after blood enters the chamber. Backflows that can be seen on the external sides of leaflets cause stagnation regions visualized in figures 4a and 5.

Regions of stagnations during diastole are visualized in figure 4. The volume of regions for which velocity is lower than 0.01 m/s is marked in colours depicted in the legend. In figure 4b only very small spots, in the vicinity of the inlet valve, are visible, which proves that the velocity in the whole domain is larger than the value mentioned in the case of the chamber with single disc valves. On the contrary, large regions in the vicinity of the inlet side valve are observed in the case of the VAD with three-leaflet valves.

Stagnation regions during systole are depicted in figure 5. Although those regions are large, it is necessary to remember that stagnation occurs in a limited time of the heart operation cycle. The most dangerous is a situation for which stagnation volumes are observed for both diastole and systole in the same regions, especially in the vicinity of the diaphragm. This conclusion has been drawn on the basis of the results of clinical observations.

The main aim of this study was to compare a disc heart valve and a polyurethane three-leaflet valve operating under the same conditions in the same chamber. Only streamlines of diastole are presented in this paper as the flow conditions are better.

It has been mentioned previously that although regions of stagnation are large for the systole conditions, they exist temporarily. Nevertheless, in the case of the three-leaflet valve, stagnation regions are significantly larger and thus the contact time of blood with the wall during one heart operation cycle elongates. It has to be pointed out that stagnation regions are visible in the back part of the chamber in the vicinity of the membrane and the blood chamber connection, which is very important and dangerous. Another very important risk factor results from the fact that there are regions of stagnation visible in systole and diastole that occur between leaflets of the valve and the adaptor wall in the case of the three-leaflet valve (compare figures 4a and 5a). This occurs only for the inlet valve. In these regions, thrombus generation is considerable. Similar results were reported for camber designed for adults (Obidowski et al. 2010).

4. CONCLUSIONS

In the presented numerical analysis, it is shown that the three-leaflet valve exhibits a high risk of blood coagulation in the place where the leaflet is attached to the adapter wall. Some significant regions of stagnation were observed for diastole and systole.

The single-disc mechanical valve inserted in the optimal angular position ensures good flow circulation during diastole and systole as well. In the case of diastole, no significant regions of stagnation were observed.

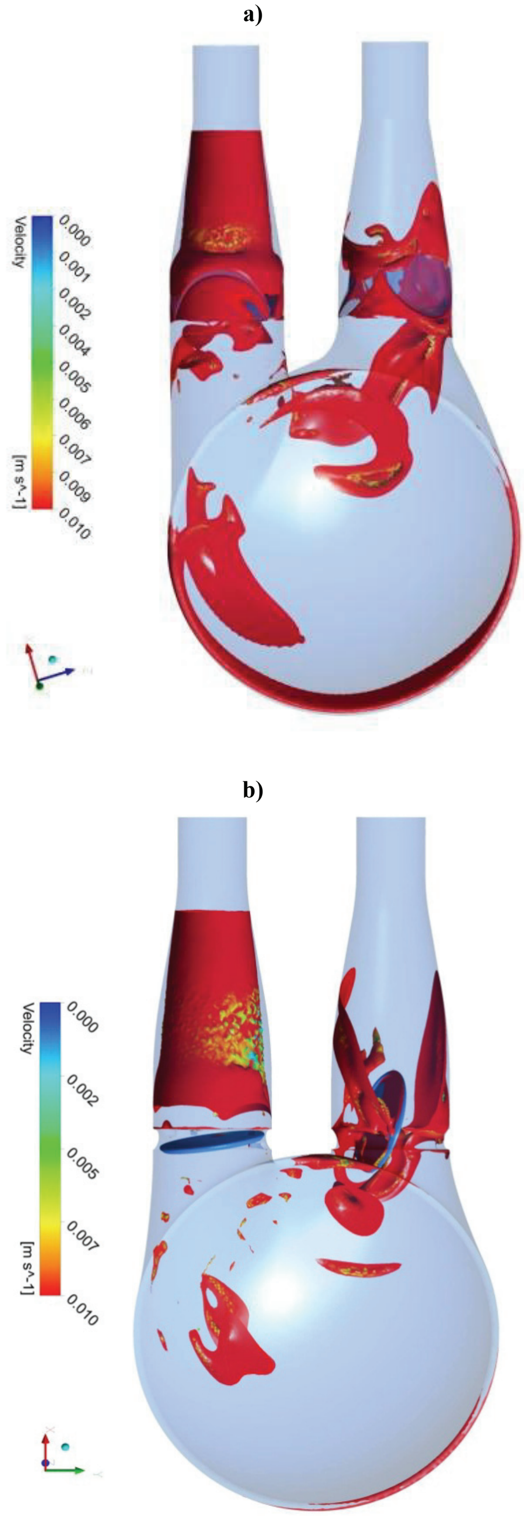


Fig. 5. Stagnation regions during systole, a) three-leaflet valve b) single disc valve.

Changes in angular positioning of the three-leaflet valve would not provide any improvement in flow conditions as the regions of stagnation are not related to the geometry of the chamber but to the design of the valve itself.

A further study of time-dependant boundary conditions, possibly involving simulations of the fluid structure interaction (De Hart et al., 2003), should be carried out to support the conclusions drawn here.

ACKNOWLEDGMENT

This work has been financially supported by the "Polish Artificial Heart" governmental project.

REFENRECES

Bujok, W., Obidowski, D., Kapis, A., Kustosz, R., Jóźwik, K., Reorowicz, P., Kłosiński, P., 2010, Influence of the type of artificial valve onto the flow inside of pediatric ventricular assist device, *Journal of Artificial Organs*, 33 (7), 471.

De Hart, J., Baaijens, F.P.T., Peters, G.W.M., Schreurs, P.J.G., 2003, A computational fluid-structure interaction analysis of a fiber-reinforced stentless aortic valve, *Journal of Biomechanics*, 36, 699–712.

Gijsen, F.J.H., Allanic, E., van de Vosse, F.N., Janssen, J.D., 1999, The Influence of the Non-Newtonian Properties of Blood on the Flow in Large Arteries: Unsteady flow in a 90° Curved Tube, *Journal of Biomechanics*, 32, 705–713.

Gijsen, F.J.H., van de Vosse, F.N., Janssen, J.D., 1999, The Influence of the Non-Newtonian Properties of Blood on the Flow in Large Arteries: Steady Flow in a Carotid Bifurcation Model, *Journal of Biomechanics*, 32, 601–608.

Johnston, B., Johnson, P., Corney, S., Kilpatrick, D., 2004, Non-Newtonian Blood Flow in Human Right Coronary Arteries: Steady State Simulation, *Journal of Biomechanics*, 37, 709-720.

Jozwik, K., Obidowski, D., Klosinski, P., 2009, Modifications of an Artificial Ventricle Assisting Heart Operation on the Basis of Numerical Methods, *Turbomachinery*, 135, 61-68.

Jozwik, K., Obidowski, D., 2010, Numerical Simulations of the Blood Flow through Vertebral Arteries *Journal of Biomechanics*, 43, 177-185.

Jóźwik, K., Obidowski, D., Reorowicz, P., Kłosiński, P., 2009, Selection of a Heart Valve Prosthesis for the Artificial Heart, *The International Journal of Artificial Organs*, Special issue dedicated to the ABSTRACTS from the XXXVI ESAO Congress, 2-5 September 2009, Compiegne, France, 25, 32 (7), 429.

Obidowski D., Klosinski P., Reorowicz P., Jozwik K., 2010, Influence of an Artificial Valve Type on the Flow in the Ventricular Assist Device, *MEDICON 2010, IFMBE Proceedings*, 29, 410–413.

Obidowski D., Mysior M., Jóźwik K., 2008, Comparison of Ultrasonic Measurement and Numerical Simulation Results of the Flow through Vertebral Arteries. In: *4th European Conf. of the International Federation for Medical and Biological Engineering*, Antwerp, Vol. 22, 286-292.

Walburn, F.J., Schneck, D.J., 1976, A constitutive equation for whole human blood, *Biorheology*, 31, 201-218.

ANALIZA WPŁYWU RODZAJU SZTUCZNEJ ZASTAWKI SERCA NA PRZEPŁYW KRWI WEWNĄTRZ KOMORY WSPOMAGANIA SERCA

Streszczenie

Urządzenia wspomagania serca (Ventricular Assist Device VAD) są używane w przypadku poważnych chorób serca, w przypadku gdy naturalne serce ma niewystarczającą wydajność pracy. Jakiekolwiek uszkodzenie lub nieprawidłowe funkcjonowanie takiego urządzenia może skutkować śmiercią pacjenta. Z tego względu prowadzone są ciągłe prace nad ich udoskonalaniem i rozwijaniem również w zakresie zastawek.

W prezentowanym artykule autorzy prezentują możliwości najnowszych metod projektowania z wykorzystaniem oprogramowania CAD oraz numerycznej dynamiki płynów. Celem pracy jest analizy przepływu przez pneumatyczną komorę serca zaprojektowaną przez Fundację Rozwoju Kardiochirurgii im. Z. Religi z Zabrza wyposażoną w dwa różne typy zastawek. Porównano pracę zastawki jednodyskowej zaprojektowanej w Instytucie Maszyn Przepływowych na bazie projektu Prof. J.J. Molla z trójpłatkową zastawką poliuretanową. Przeprowadzono analizę na podstawie wizualizacji przepływu wewnątrz komory krwistej sztucznego serca. Zilustrowane zostały obszary stagnacji występujące w komorze, w których może dojść do wykrzepiania krwi. Położenie kątowe w kanałach dolotowych i wylotowych w przypadku zastawki jednodyskowej zostały wytypowane na podstawie wcześniej prezentowanych prac. Przeprowadzono obliczenia z założeniem przepływu ustalonego, nieodkształcalnych ścian komór, konektorów i zastawek. Lepkość dynamiczna krwi zdefiniowano na bazie modelu wykładniczego, dzięki czemu oddano nienewtonowski charakter krwi. Porównania dokonano dla systoli i diastoli. Do wykonania wszystkich faz obliczeń: przygotowania, rozwiązania i opracowania prezentowanych wyników użyto komercyjnego kodu Ansys CFX v12. Odkształcenie zastawki trójpłatkowej w fazie otwarcia uzyskano z pomocą oprogramowania SolidWorks 2009.

Na podstawie dokonanych analiz wykazano, że jednodyskowa mechaniczna zastawka serca generuje lepsze warunki przepływu w komorze. Jest to szczególnie widoczne gdy analizie poddaje się obszary stagnacji i ryzyko wyrzepienia krwi. Ponadto zastawka dyskowa umożliwia ukierunkowanie przepływu, co powoduje bardziej jednorodny przepływ umożliwiający dobre wypłukiwanie krwi z miejsca połączenia komory z membraną.

Received: September 30, 2010
Received in a revised form: December 16, 2010
Accepted: December 18, 2010