Publishing House
AKAPIT

COMPUTER METHODS IN MATERIALS SCIENCE
Informatyka w Technologii Materiałów

Vol. 11, 2011, No. 1

NUMERICAL FLOW ANALYSIS IN POLVAD-IMPL PROSTHESIS MODEL

WOJCIECH BUJOK*, MACIEJ DARŁAK, DOROTA JURKOJĆ, ROMAN KUSTOSZ

Foundation for Cardiac Surgery Development, Wolności 345a, 41-800 Zabrze, Poland
**Corresponding author: wbujok@frk.pl*

Abstract

Numerical analyses of flow properties for POLVAD-IMPL device equipped with Medtronic HallTM valves were performed. Steady flow conditions were assumed in all calculations. Analysis concerned the device in full filling position, with inlet and outlet valves open and steady determined flow rate was performed. The analysis of inlet angular valve position in the prosthesis connector in full filling state and outlet valve nearly closed was also performed. Mesh sensitivity analysis on calculation results was examined.

Comparative analysis of angular valve position in implantable VAD construction, as well as the analysis of the construction itself, showed that it is necessary to modify the geometry due to hazardous fluid stagnation areas in blood chamber volume and irregular outflow from the chamber into the outlet connector. The construction with angular valve position of 90^{o} (relative to the base) was characterized by the best flow conditions. However advanced analysis showed significant mesh influence on stagnation areas in the chamber. Better detection of stagnation areas was probably reached due to better mesh “resolution” in the wall vicinity. Despite of multi-million element meshes resulting with significant extension of calculation time, performing analyses with different variants of meshes seems to be important. There is a question constantly where is the “good” mesh limit and how to determine, when it is reached.

Key words: numerical simulation, flow analysis, ventricular assist device, mesh sensitivity analysis, CFD

1. INTRODUCTION

Construction of implantable ventricular assist device POLVAD-IMPL was developed within the Polish Artificial Heart Program. Results presented below are part of POLVAD-IMPL construction development and concerns the analysis of the device selected version.

2. OBJECTIVES

Numerical flow analysis of the developed implantable assist device was done to study the construction and potentially dangerous areas localization, to investigate where blood clot formation could happen. Numerical modeling helped with the designing process of POLVAD-IMPL ventricular assist device.

3. MATERIALS AND METHODS

Numerical analysis of flow properties was carried out in ANSYS CFX. The calculations were done pursuant to the VAD geometry model designed in Artificial Heart Laboratory. Implantable VAD model geometry, was discretizated in Ansys ICEM CFD.

First the VAD analysis with the steady flow was performed in the full filling phase (diastola) with both valves open (inlet and outlet). Then, several studies concerning the inlet valve angular position (turning angle 45°) in the same membrane position (diastola) were carried out. The analysis of VAD

ISSN 1641-8581

model was executed with constant flow velocity 0.6 m/s on the inlet (average flow in the chamber was about 4 l/min). In order to stabilize the flow conditions in the VAD chamber, inflow/outflow canals were added (100 [mm] length ≈ 8 inlet/outlet connector diameter multiple) (Bujok et al, 2009). At the VAD chamber outlet constant pressure about 13 kPa was assumed (typical human vascular systemic load). The reference pressure was set at 100 kPa. The gravity forces were ignored. The Newtonian model of blood was used for the calculations, the blood density 1060 kg/m^3 was assumed. The adopted value of dynamic viscosity was 0.0034 Pa·s, the SST turbulence model (Shear Stress Transport) was used (Johnston et al., 2004; Jóźwik & Obidowski, 2010). Calculations were performed assuming undeformable chamber walls and valves. The outlet valve was not fully closed in order to perform the computational analysis of the inlet valve angular position. Incomplete valve closure allowed fluid flow around the disc (the continuity condition of the mesh). However, the fluid velocity in this area (narrow gap), was significantly overrated in relation to reality. The reason was steady state flow simulation assumed. For all cases, the computational domain was determined by VAD volume projecting the blood chamber interior with the inlet and outlet connectors, where the valve discs were situated (Obidowski et al, 2010). The pre-analysis conditions were shown in figure 1.

Numerical mesh models were created in ANSYS ICEM CFD. Hybrid unstructured meshes were prepared, consisting of tetrahedral elements in domain centre and prismatic elements in the boundary layer around the limiting domain discs and the valves rings surface. The total number of mesh elements in each simulation was defined about 0.8 million.

In order to analyze the sensitivity of model mesh density, unstructured mesh was built consisting of 5 million elements (treated as reference mesh). Due to long computational time, the calculations were limited only to the model with two open valves (steady state flow conditions in the chamber). Figure 2 presents examples of mesh elements.

For each case, the mesh quality tests were performed in ANSYS ICEM CFD software during its creation, as well as after the flow calculations within the dimensionless parameter y^+. In all meshes used in the calculations, the y^+ parameter did not exceed the SST turbulence model limit, which amounts 20 (Jóźwik et al, 2009; Obidowski et al, 2010). The best y^+ value was obtained for reference mesh (5 million elements, figures - indication ZOG), and it was equal to 2.37 (figure 3). For VAD with opened valves (indications ZO) y^+ value 6.11 was obtained. The worst y^+ was equal 17.66 for VAD mesh, where the inlet valve angle was turned 135° (the base valves angle position like in the ZO VAD).

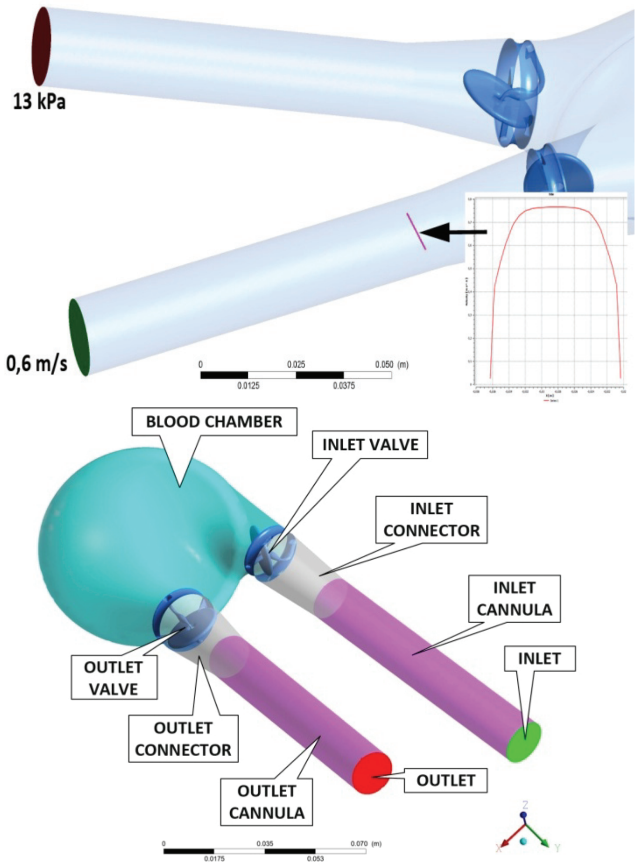


Fig. 1. Boundary conditions and computational domain of the VAD with opened valves (the base position).

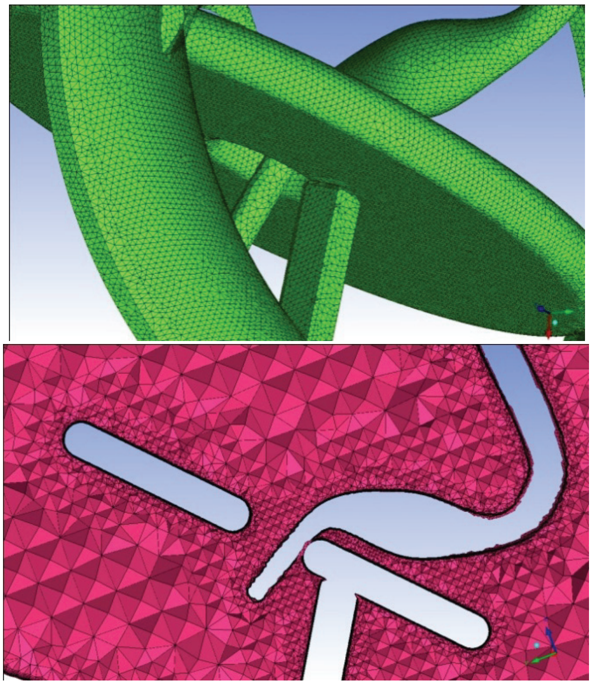


Fig. 2. Exemplary numerical VAD mesh (valves view).

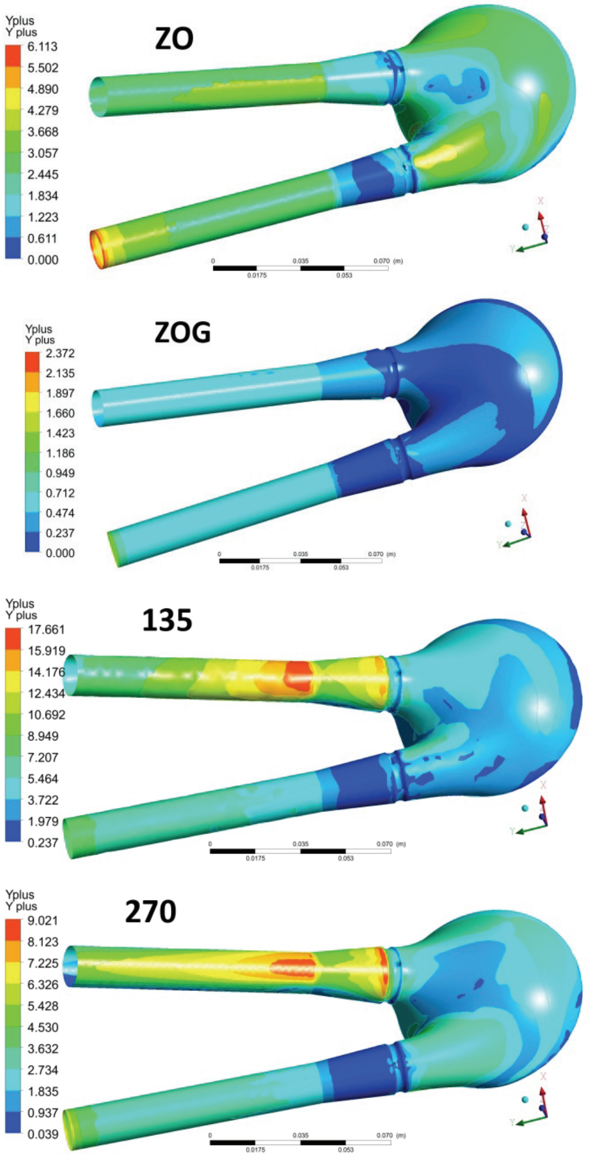


Fig. 3. *Dimensionless y^+ parameter for VAD meshes: ZO – valves opened; ZOG – valves opened, reference mesh; 135, 270 – selected inlet valve angle position, outlet valve nearly closed.*

The evaluation of the solution progress was done pursuant to maximum and average values graphs of the following iteration residuum. The flow character influenced strongly the convergence levels achieved for VAD numerical models. Due to the strong whirling flow character, the level of convergence was below 1e-3 RMS, despite the fact of continuation the calculations for the next several hundred iterations.

4. RESULTS

The following figures show the flow structure in the blood chamber of VAD. Figure 4 shows the velocity vectors in cross-section chamber planes with opened valves. On the left side the sections were performed for 0.8 million element mesh, on the right side for the reference mesh. The number of mesh elements, has no significant effect on the flow structure in the VAD chamber. Analyzing the figures, it can be concluded that, there are large areas of relatively low velocity vectors in the chamber center. Fluid coming into the chamber from the inlet connector, despite orientation caused by the open valve, does not wash the whole blood chamber sidewall (on the right side from the channel axis). Hence, the areas of low velocity values are created there, the fluid turns back creating the vortexes there, shown in figure 5. The farthest area in the blood chamber, which is located opposite to the connectors, is also poorly washed.

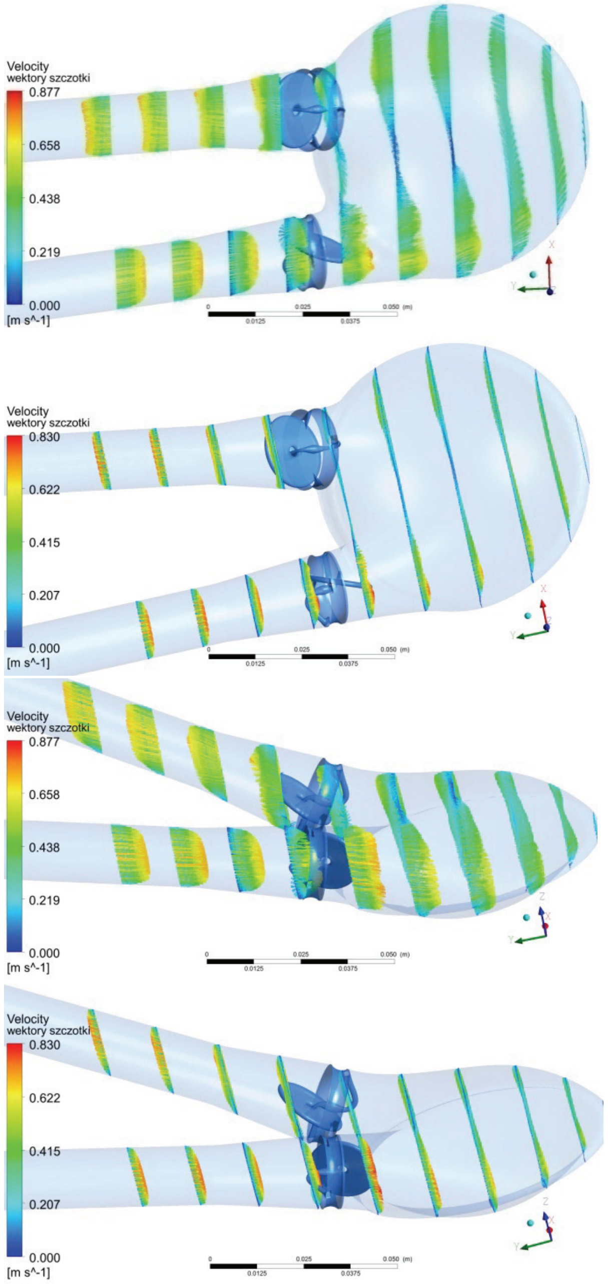


Fig. 4. *Velocity vectors in cross-section planes of blood chamber for ZO mesh and ZOG mesh.*

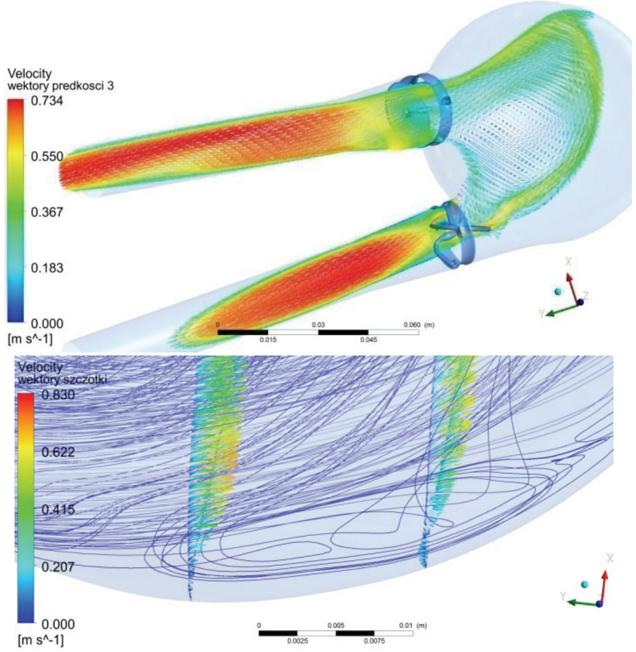
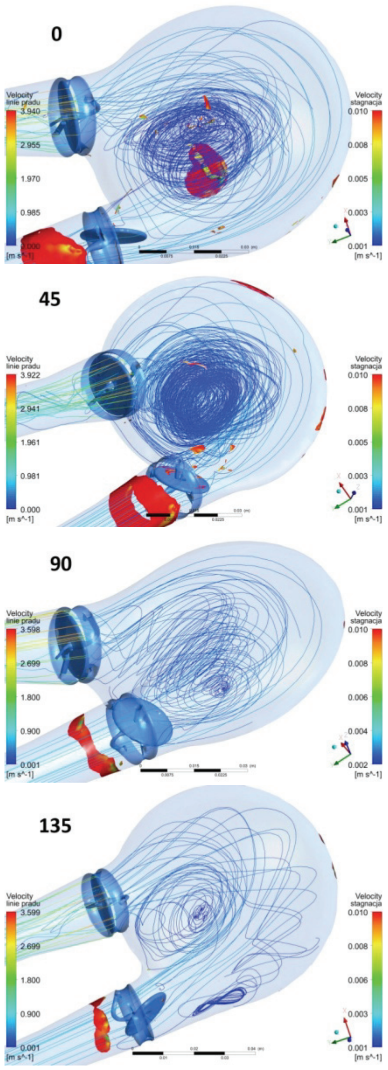


Fig. 5. Velocity contour distribution in the longitudinal plane and whirls in blood chamber of VAD.



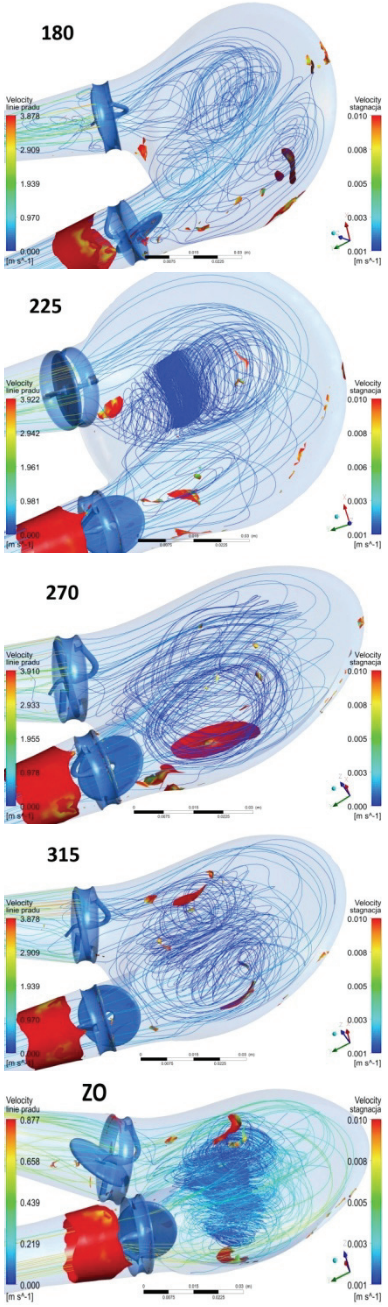


Fig. 6. Streamlines and the areas of stagnation for the blood chamber with the following inlet angular valve position and for the VAD with open valves (ZO).

The most regular flow occurs in VAD with the angular valve position 90°. However, in each analyzed case, swirl turbulence in the outlet connector can be observed. In each analyzed case the stagnation zones can be also observed. The stagnation zones are the potential places of blood clotting formation in the VAD chamber. However, definitely the smallest stagnation zones occur for the angular valve position 90°. The worst flow conditions are obtained for the angular valve positions for more than 135°. The huge irregular whirls in the blood flow in VAD chamber appear, which will complicate the complete volume emptying, which is shown in figure 6.

The flow structure in VAD with two valves in maximal open position was also analyzed (continuous flow through the blood chamber under the steady state conditions). The analyses showed a spiral vortex in the center and the stagnation zones appearing in the blood chamber region behind the inlet connector. Large stagnation areas are also observed in the inlet connector before the valve, the smallest at the chamber top and the diaphragm, presented in figure 6.

As it was concluded above, the number of elements in mesh has no significant effect on the flow conditions in the VAD chamber. However, the mesh with more elements located in the boundary layers (prismatic elements) showed essentially larger areas of stagnation (Jóźwik et al, 2009; Obidowski et al, 2010) which can be seen in figure 7. The not previously detected flow stagnation areas, are located in: membrane corner, outlet connector and at outlet valve.

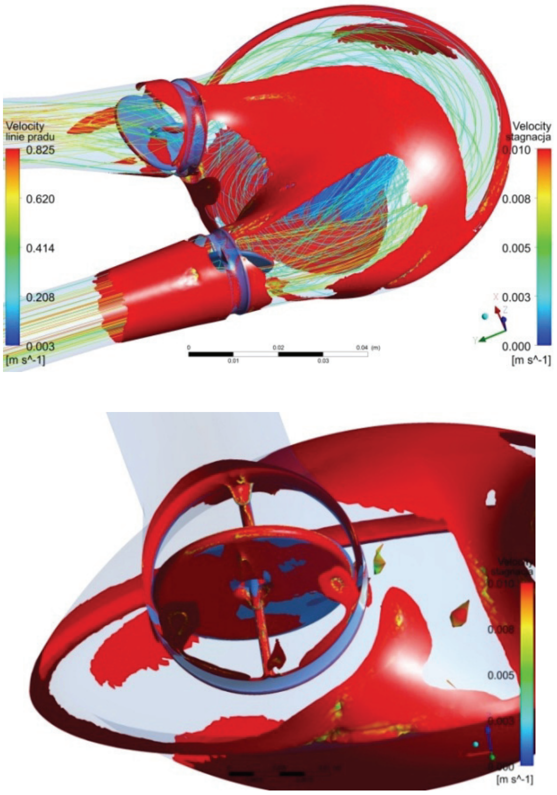


Fig. 7. Streamlines and the areas of flow stagnation for the reference mesh (ZOG) in the blood chamber.

The analysis showed that in the VAD construction, despite the changes of the inlet angular valve position, a lot of whirls and large areas of stagnation are formulated. The best results are obtained for inlet angular valve position 90°. However, mesh sensitivity analysis revealed that the mesh quality is very important (the number of elements, especially prismatic) for detecting the zones of stagnation in the blood chamber.

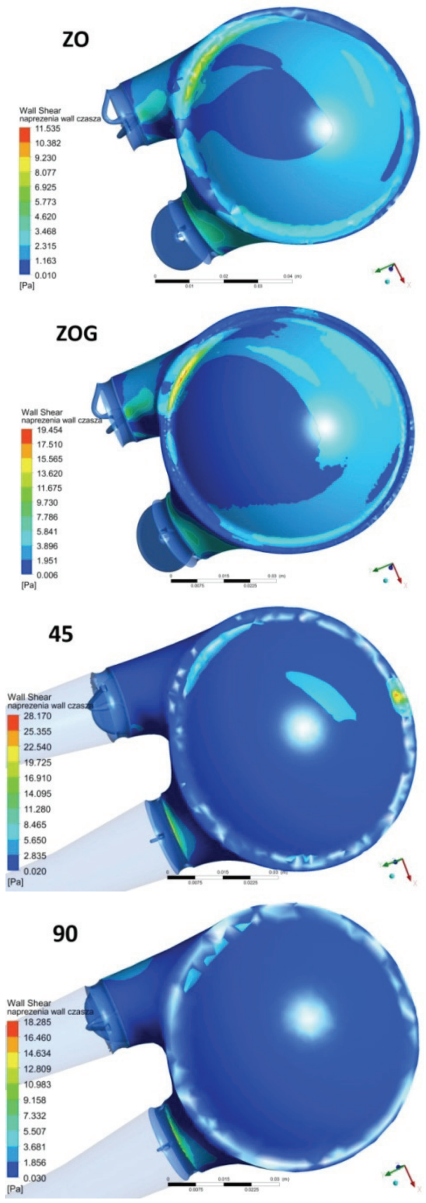


Fig. 8. The wall shear stress in blood chamber.

The wall shear stress in the blood chamber are presented in figure 8. In none of the analyzed cases, the stress values do not exceed a value 100 Pa (for all calculated angular valve position, the values fluctuated in the range 17÷28 Pa). However, it has to be noted that the calculated values in ANSYS CFX, strongly depend on mesh quality, the local values of velocity and the object geometry. For ZO mesh, the maximum stress in the blood chamber is 11.54 Pa, while for the same mesh structure ZOG, the maximum is 19.45 Pa. Hence, while considering these values, is necessary to remember that without the experimental data of each construction, stress value reliable verification is difficult.

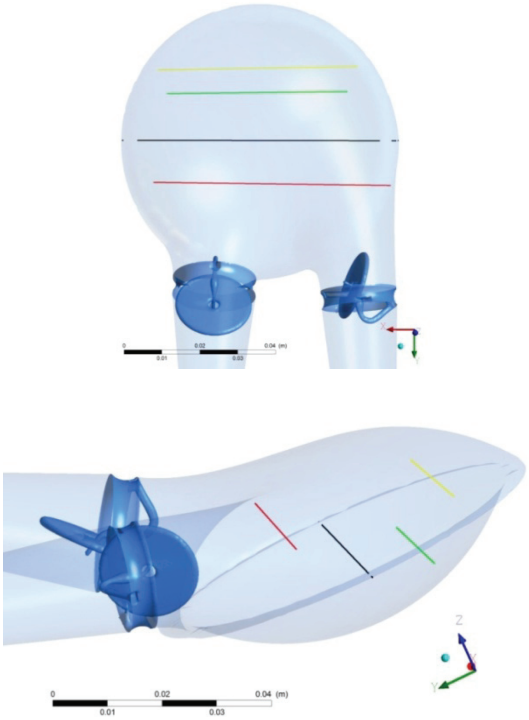


Fig. 9. Cut lines of velocity profiles in the blood chamber (top and side view).

The following figures present the velocity profiles in the selected areas in blood chamber. Figure 9 shows the exemplary cut lines, along which the velocity values were sampled.

The velocity profiles for two different meshes of the same VAD version (with the open valves) are presented in figure 10. Good agreement of the flow character, in spite of the mesh used for calculations is observed.

5. DISCUSSION AND CONCLUSIONS

Mesh independence test showed no significant differences in the parameters and the flow character in the same calculation model (blood chamber with open valves in the base position). It confirms that there is no generated mesh influence on the calculation results. However, the mesh with bigger number of elements (the boundary layer from prismatic elements was about 2 million element big), showed significantly larger areas of stagnation in the blood chamber, when the other meshes were used (with fewer number of elements).

Comparative analysis of the angular valves position (Jóźwik et al, 2009) in the construction of the implantable ventricular assist device, as well as the analysis of the VAD construction itself, revealed that it is necessary to eliminate the dangerous zones

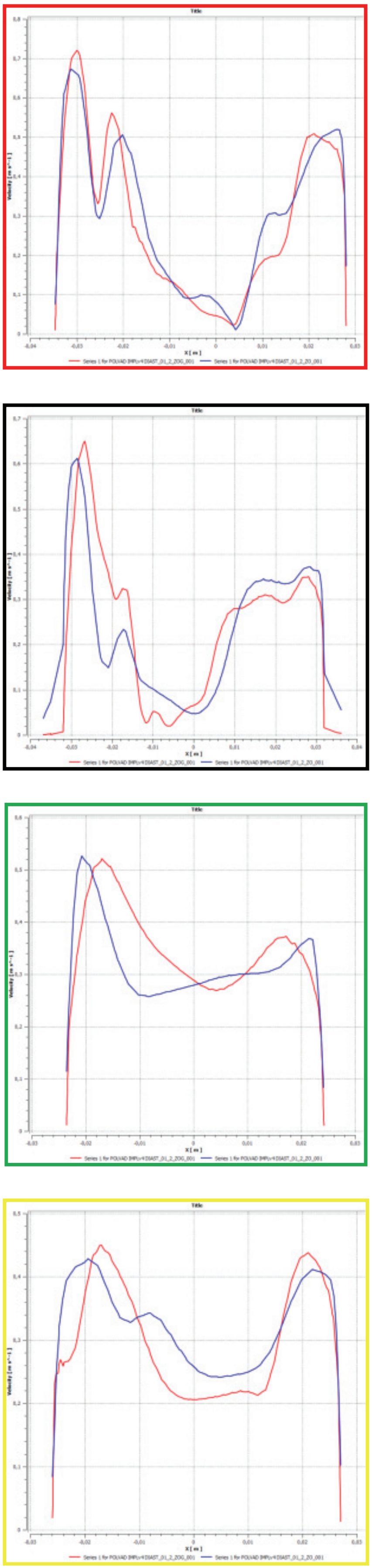


Fig. 10. Velocity profiles at chosen cut lines of the blood chamber with the valves in open position (in red - ZO mesh) and for the blood chamber with the reference mesh (in blue - ZOG mesh).

of stagnation in the blood chamber, and to improve the outflow from the blood chamber to the outlet connector. There is a fluid swirl in the outlet connector. Probably it is caused by the outlet angular valve position, which was not analyzed in this work. It is more likely, that the fluid swirl arises from the big whirl formed in the region between the connectors in the blood chamber.

The construction with angular valve position of 90° (relative to the base) was characterized by the best flow conditions. However, mesh sensitivity analysis showed a large mesh quality influence on the stagnation areas in the blood chamber. Better detection of stagnation areas was probably reached due to better mesh "resolution" in the boundary layers. Despite of multi-million element meshes resulting with significant extension of calculation time, performing analyses with different variants of meshes seems to be important. There is a question constantly where is the "good" mesh limit and how to determine, when it is reached. Whether 5 million element mesh is sufficient enough to project the flow through the investigated object, or it is a 16 million element mesh or bigger. So, the next question becomes reasonable, whether the method of determining the regions of flow stagnation in the blood chamber, by the way of assigning the surfaces where the velocity is lower than 0.01 m/s is correct (Jóźwik et al, 2009; Obidowski et al, 2010). A further study for time-dependant boundary conditions and possibly involving fluid structure interaction simulations (De Hart et at., 2003) should be carried out to support the conclusions drawn here.

REFERENCES

Bujok, W., Jurkojć, D., Szuber, A., Kustosz, R., 2009, Analysis of the new disc valve construction using numerical flow modeling, *Journal of Artificial Organs*, 32 (7), 428.

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.

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.

Jóźwik, K., Obidowski, D., 2010, Numerical simulations of the blood flow through vertebral arteries, *Journal of Biomechanics*, 43 (2), 177-185.

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

Obidowski, D., Kłosinski, P., Reorowicz, P., Jóźwik, K., 2010, Influence of an Artificial Valve Type on the Flow in the Ventricular Assist Device, *Medicon 2010*, IFMBE Proceedings, 29, 410-413.

ANALIZA NUMERYCZNA PRZEPŁYWU W IMPLANTOWALNEJ KOMORZE WSPOMAGANIA SERCA POLVAD IMPL

Streszczenie

W ramach jednego z zadań Programu Polskie Sztuczne Serce, opracowano konstrukcję nowej implantowalnej komory wspomagania serca POLVAD-IMPL. Rezultaty prezentowane w artykule, są jednym z etapów realizowanych prac dla opracowania nowej konstrukcji komory i dotyczą analizy jednej z wersji opracowywanej komory POLVAD IMPL. Analiza numeryczna przepływu ma na celu zbadanie samej konstrukcji oraz wyłonienie potencjalnie niebezpiecznych obszarów, w których może wykrzepiać krew. Modelowanie numeryczne wspomaga proces projektowy komory POLVAD-IMPL. Symulacje zostały przeprowadzone w warunkach ustalonych dla komory z zastawkami Medtronic HallTM. Analiza została przeprowadzona dla komory w fazie pełnego napełnienia przy maksymalnym otwarciu zastawek (wlotowej i wylotowej), dla stałego, ustalonego przepływu. Przeprowadzono również analizę ustawienia kątowego zastawki wlotowej w konektorze napływowym komory w pozycji pełnego napełnienia z domknięta zastawką wylotową. Wykonano analizę wrażliwości siatki na wyniki rozwiązania.

Received: November 8, 2010
Received in a revised form: November 16, 2010
Accepted: November 17, 2010