flow analysis within artificial heart valve,
numerical flow analysis, CFD

Tomasz MOSZKOWSKI[1], Michał JAWOREK[1], Benita KOSTRZEWA[1], Krzysztof LALIK[2],
Maciej DARŁAK[3], Ievgenii ALTYNTSEV[3], Roman KUSTOSZ[3]

FLOW ANALYSIS WITHIN MECHANICAL HEART VALVE – MEDTRONIC HALL - AND VALIDATION OF RESULTS BY NUMERICAL MODELLING

Research was conducted to analyze the flow of a fluid within mechanical heart valve - Medtronic Hall. Physical experiment and numerical modelling were performed. The aim of the research was to determine the difference between obtained experimental and numerical data.

In the experiment a dependency between static flow rate within the valve and static inlet and outlet pressure in the valve duct was examined. Moreover a dependency between static flow rate and angular valve position was also determined.

Experimental data was used to perform a numerical flow analysis. The obtained flow rate values and angular positions of the valve were set to a finite-volumes-method model in order to achieve model output pressure values identical or similar to the ones obtained from the experiment.

The resulting pressure values from the experiment and numerical analyses proved to be of the same order of magnitude, varying only by up to 10%. However, as far as differential pressure is concerned, numerical results were out of the range of measurement resolution. It can be assumed that numerical flow analyses quite correctly predict the real phenomenon and in view of measurement inaccuracy of used sensors authors would suggest using more accurate ones and repeating measurements for future clarification.

1. INTRODUCTION

Usually numerical modelling is performed to predict and visualize behaviour of a real systems based on their models [1]. Conducted research used this method for slightly different purposes – the numerical modelling was used as a validation tool. Physical experiment was performed and on the basis of this experiment's outcomes numerical analyses were made.

The aim of the research was to compare results of the physical experiment and numerical modelling, where both were conducted to analyze the flow within a mechanical heart valve Medtronic Hall. Static flow rate, static pressure at the inlet and outlet of the valve and angular valve position were examined. The authors aspired to calculate the difference between outcomes from both used methods.

The following experimental data - static flow rate within the valve, static outlet pressure and an angular inclination of the valve's disc caused by the flow - was used in the numerical flow analyses. The simulations were done to calculate inlet and outlet pressure, as well as deliver a flow visualization within the valve.

2. METHODS

This section contains methods used in the physical experiment and numerical modelling.

2.1. PHYSICAL EXPERIMENT

In order to carry out a proper experiment, a working place should be prepared as well as a research algorithm had to be followed. The following section describes both of the aforementioned aspects.

[1] AGH University of Science and Technology, Multidisciplinary School of Engineering in Biomedicine, Mickiewicza 30, 30-059 Krakow, Poland.
[2] AGH University of Science and Technology, Department of Process Control, Mickiewicza 30, 30-059 Krakow, Poland.
[3] Foundation of Cardiac Surgery Development, Artificial Heart Laboratory, Wolności 345a, 41-800 Zabrze, Poland.

A measuring stand was prepared to perform a series of experiments. Figure 1 shows it's schema. It consists of: a pump (A), an ultrasonic flowmeter FLEXUS® AMD (B) (accuracy 1-3%), two Peltron® pressure sensors installed at inlet (E1) and outlet (E2) of the valve (about 5 cm far from the valve) with 0.25% accuracy of the sensors' range of ±100 kPa, a high resolution digital camera (G) focused on the valve's side (in order to obtain angular valve position). Mechanical heart valve Medtronic Hall was fixed permanently into a plastic tube (F). The duct was comprised of a range of plastic tubes, a on-off valve (D) and an equalizing tank (C). Water was used as a fluid in the experiment.

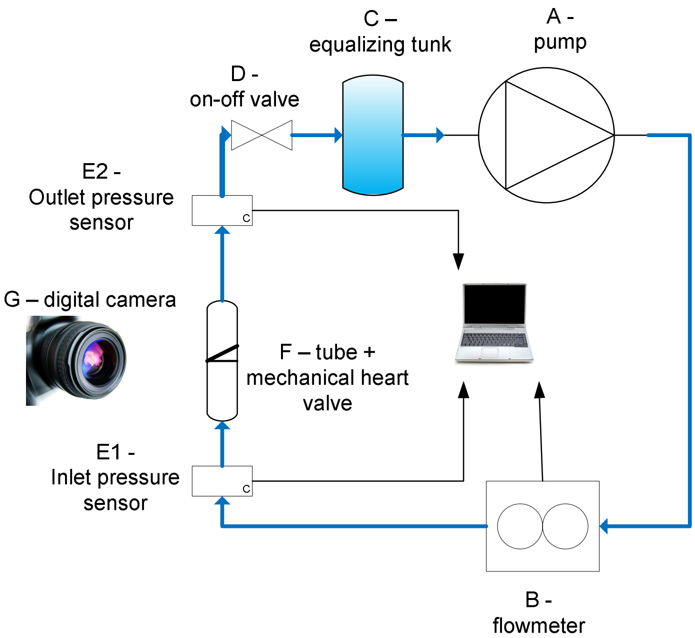

Fig. 1. Schema of experimental stand.

During the experiment different rates of the flow were set which resulted in changes in valve angular position and in pressure values. Each time the stable flow rate and pressures at the inlet and outlet, corresponding to a specific flow rate, were measured. A series of 5 pictures were taken while analyzed flow rate was stable, so that the angle between valve's ring and valve's disc could be captured, measured using graphical software and averaged afterwards. For all measurements statistical analyses were performed.

2.2. NUMERICAL MODELLING

In order to correlate the inlet and outlet pressure as well as the flow of the liquid medium with the opening angle of the valve a numerical simulation using finite volumes method was used, aside of the experimental data [2,3]. In this section one will find the basic assumptions made concerning the model as well as the schematic used in modelling and the quantities obtained.

In order to simplify the process of modelling the following assumptions were made:

1. The character of the flow of liquid is time-invariable,
2. The velocity of a liquid touching the surfaces of the valve and the duct was equal to 0,
3. The liquid used in the modelling was a Newtonian model of water at 20 deg. C (density: 997 kg/m3, dynamic viscosity: 8.899×10-4Pa·s),
4. The disc of the heart valve was mechanically fixed at a particular angle,
5. Liquid flow was introduced to the inlet of the model and outlet pressure was applied.

In total, four numerical models were created, each with the heart valve opened to a different angle value, obtained from the physical experiment, (25.6, 47.5, 52.1, 52.4 deg.). During the preparation process the same scheme was followed:

1. Preparation of a assembly drawing of the heart valve and the duct using CAD (Fig. 2),
2. Export of the drawing to FVM software and correction of geometric errors,
3. Generation and refining of a volumetric mesh (Fig. 3),
4. Introduction of material characteristics and boundary conditions,
5. Preparation of the prismatic layer in the vicinity of the walls,
6. Calculations and post-processing.

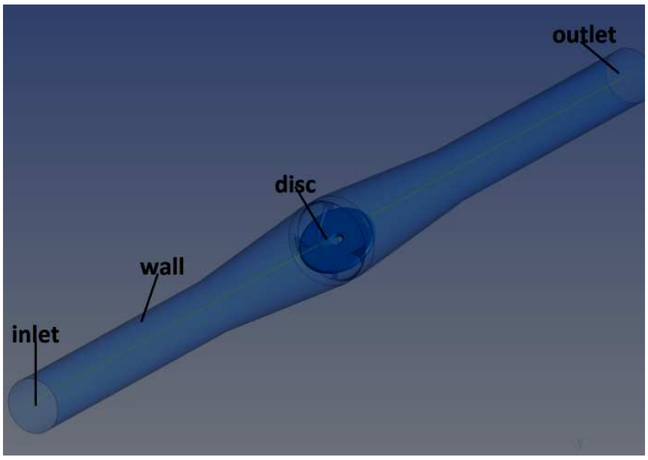

Fig. 2. Model of duct and valve.

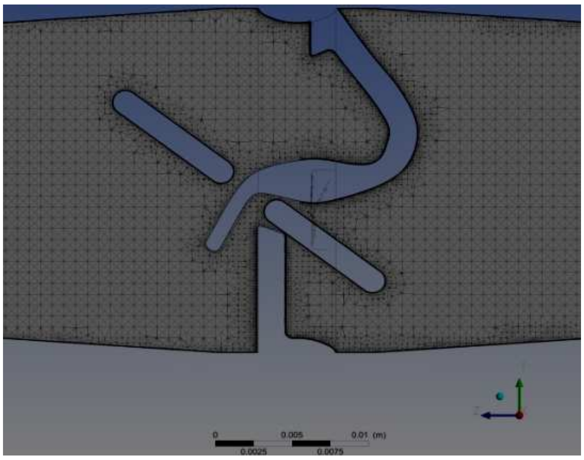

Fig. 3. A cross-sectional sample of generated meshes.

As a result of the modelling process the following physical quantities were acquired: absolute pressure field, velocity field of the liquid, differences in pressure along a specified axis of the model (OZ), static pressure in two points representing the exact place of the pressure gauges.

3. RESULTS

The following information consists of the obtained results concerning both the experiment and the numerical modelling.

3.1. PHYSICAL EXPERIMENT

Physical experiments were performed to determine the characteristics of the valve: valve opening angle as well as inlet and outlet pressure in relation to the flow rate.

Figure 4 shows the dependency of the angular valve position on the flow rate (flow rate range 0-2.2 l/min). Near linear relationship between angular valve position and flow rate for analyzed flow rate range has been observed.

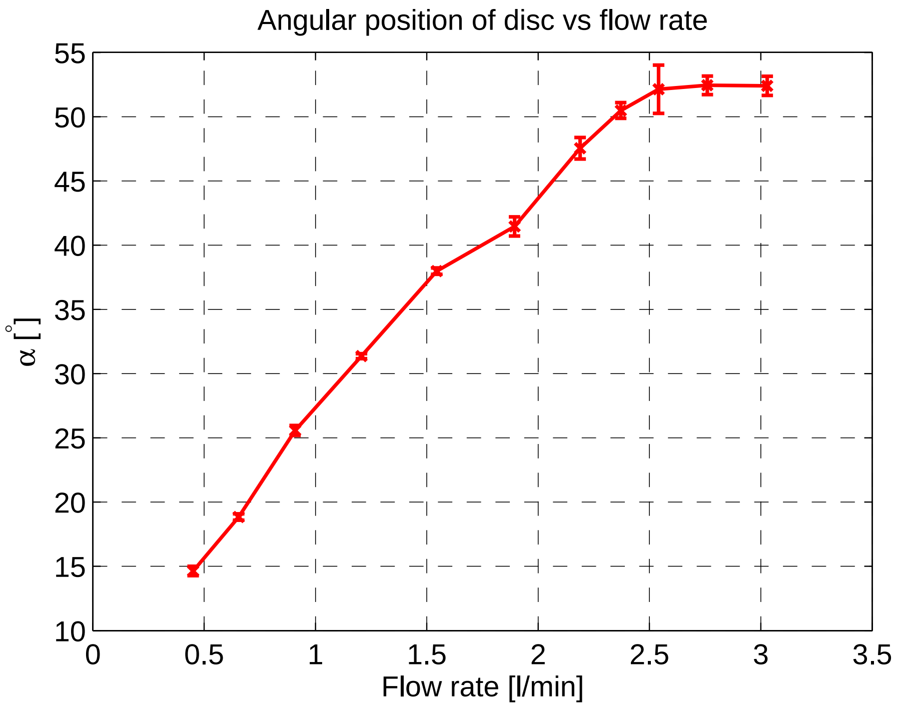

Fig. 4. Dependency of valve opening angle on the flow rate.

The maximum angle was 52.40 degrees for the flow rate of 2.76 l/min. Increasing the flow rate beyond this point has no further effect on the opening angle, however, increased turbulence was observed resulting in angle fluctuations.

The inlet and outlet squared pressure increased with increasing flow as can it be observed in Figure 5a. Inlet pressure is always higher than the outlet pressure. The pressure difference is shown in Figure 5b. Due to the value of the pressure differences being of the similar magnitude as the measurement error of the pressure gauge, it is impossible to determine other relationships between them. In addition, pressure was determined at a finite number of measured points, which for turbulent flow does not give accurate results.

Figure 5b shows that the higher flow rate causes the higher standard deviations of measured pressures. Large fluctuations in the pressure measurement points may be explained with turbulent flow in higher flow rate range. It can be assumed that with the enlarged flow the pressure difference is also increased, but the study didn't demonstrate this properly. It is why the study should be repeated using more accurate pressure transducers and additionally, with the measurement of the circumferential pressure.

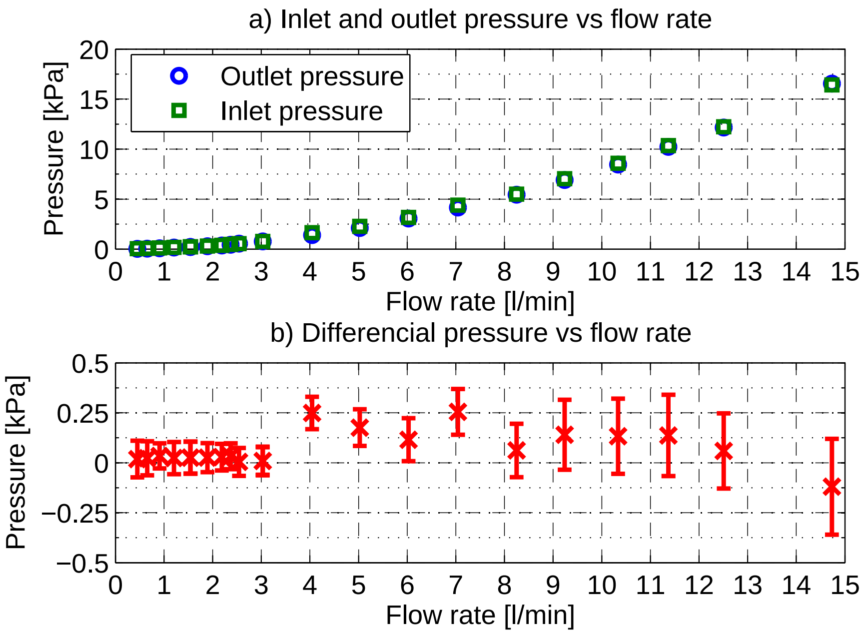

Fig. 5. a) Dependency of inlet and outlet pressure on flow rate; b) Dependency of differential pressure (difference between inlet and outlet pressure) on flow rate.

3.2. NUMERICAL FLOW ANALYSES

The following description contains results from numerical models such us: the liquid flow rate, pressure fields and absolute pressure along the OZ axis.

3.2.1. FLOW OF THE LIQUID

Figure 6 represents one of the obtained streamlines of the liquid inside the duct. One can observe turbulence as well as decrease of velocity of the liquid just in front of the disc of the valve in the inlet part of the model. Afterwards, the liquid close to the crevices accelerates to a total of 0.3 m/s velocity and decelerates to approximately 0.1 m/s.

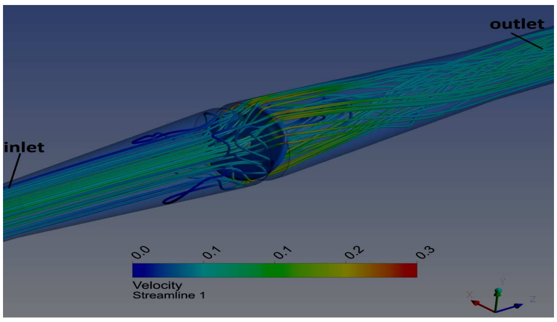

Fig. 6. Streamlines within the valve at flow rate = 0.9 L/min and angular valve position = 25.6 deg.

3.2.2. PRESSURE FIELDS

Figure 7 shows four static pressure fields in the cross-section of the flow duct, each corresponding to a different flow rate of the liquid. Also, differences in the angular inclination of the valve are observed. These changes the overall stream values of the liquid near the valve cavity. Thus, the higher the flow rate is set, the higher the pressure values are obtained.

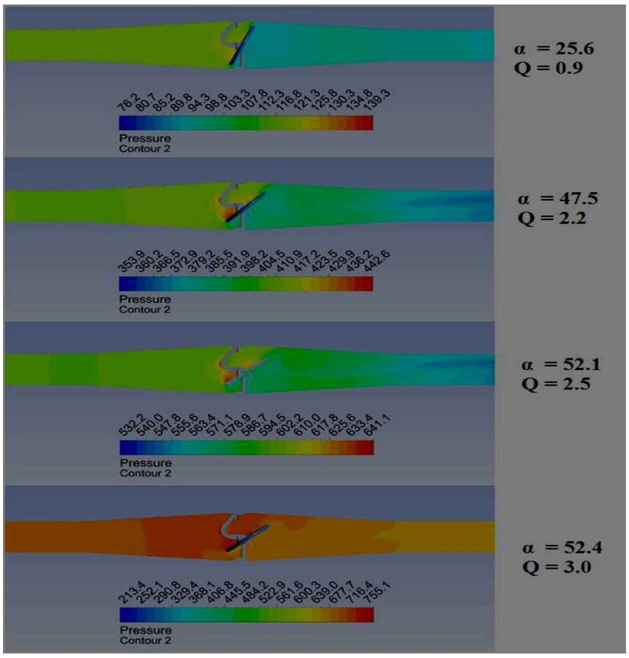

Fig. 7. Static pressure field, depending on the flow rate. The pressure is given in Pa, α[°] – valve opening angle, Q [l /min] –flow rate.

3.2.3. PRESSURE VALUES ALONG THE DUCT

Figure 8 shows the comparison of pressure measured along the duct depending on the set flow rates and the angular valve position. The higher the flow rate is, the higher the pressure in the duct is. Differential pressure also increases.

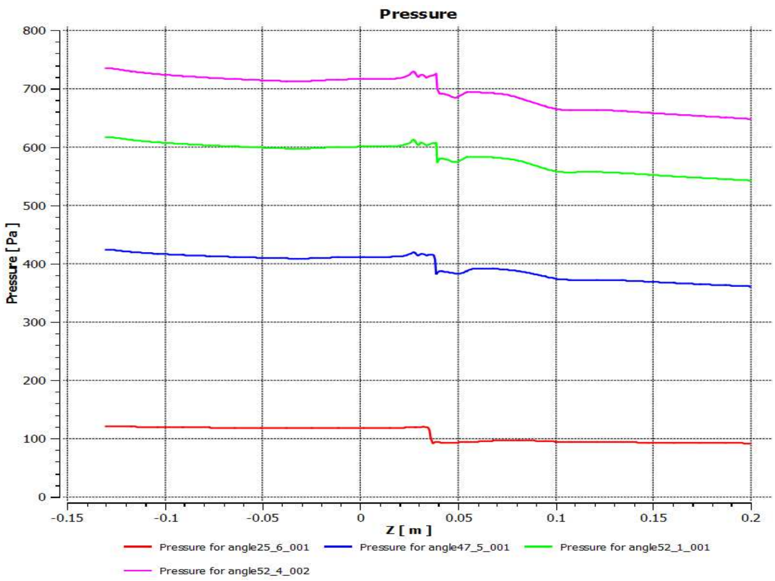

Fig. 8. Change of pressure along the duct depending on flow rate.

4. DISCUSSION

The results obtained in experimental and numerical studies were compared. As it can be seen from Table 1, numerical and experimental results are comparable to the nearest order of magnitude [i.e. > 0.1 x]. Pressure difference of experimental and modelled data vary in range of 7-57 Pa for the pressure at the inlet of the valve and in range 1-107 Pa for the pressure at the outlet. The simulation proved that a greater flow rate indicates greater pressure difference. In the experiment the pressure difference seems to be decreasing with higher flow rate, however it is only because of the specific selection of analyzing flow rate values. Measured differential pressure at inlet and outlet of valve at whole range was presented in figure 9. In order to improve the quality of the results, as it was previously mentioned, pressure transducers of greater precision should be used.

Table 1. Comparison of obtained numerical and experimental data; where: Q – flow rate, α – valve opening angle; ΔP_in- inlet pressure difference; ΔP_out- outlet pressure difference.

s				experimental data	numerical data
Q [l/min]	**α [^{O}]**	**ΔP_in [hPa]**	**ΔP_out [hPa]**	**Log10**	**Log10**
0.91	25.6	0.007	0.001	-0.925	-0.898
2.19	47.5	0.023	0.008	-0.385	-0.410
2.54	52.1	0.055	0.010	-0.220	-0.262
3.03	52.4	0.057	0.108	-0.144	-0.111

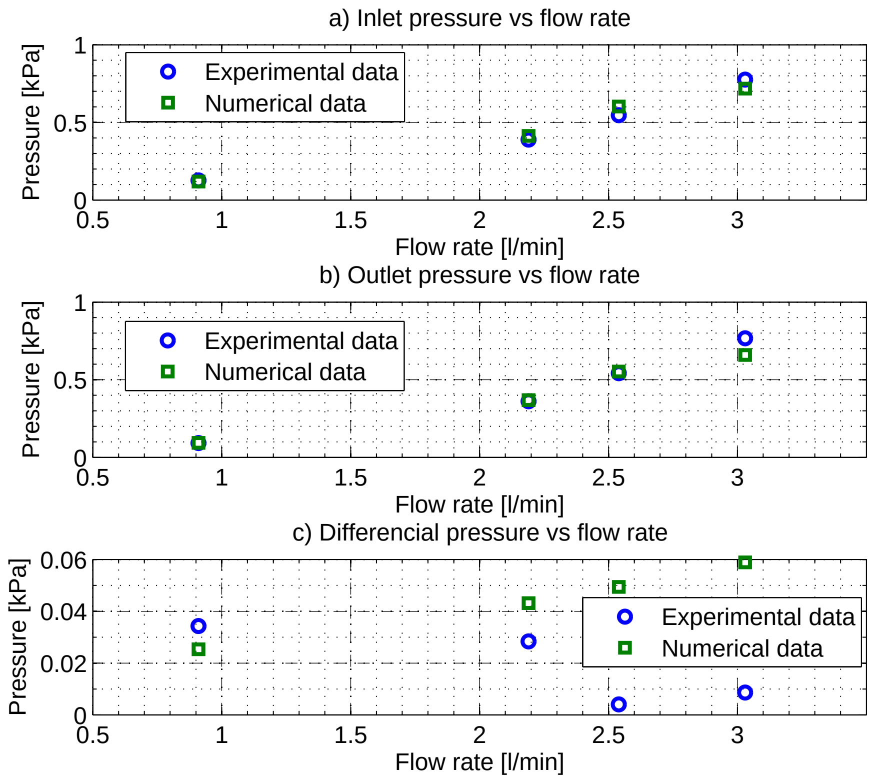

Fig. 9. a) Outlet pressure in relation to flow rate in numerical and physical experiment; b) Analogous relationship concerning the inlet pressures; c) changes in differential pressure (inlet pressure - outlet pressure) depending on the flow rate in the numerical model and physical experiment.

Figure 10 shows the summary comparison between the data elaborated from the physical experiments and from the numerical modelling. It represents the percentage difference of pressures obtained in experimental and numerical studies. Percentage difference of inlet pressure was in range of 6-10%. Due to the fact that the outlet pressure value obtained from the experiment was set as one of the boundary conditions, the pressure levels in the outlet part of the model are more consistent with the ones from the experiment. The difference of outlet pressure was in range of 2% for 3 analyzed flow rates. 10% difference of outlet pressure was noted for the largest analyzed flow rate (3.03 l/min). It was probably caused by the fact that the measurement was taken from a specific point, which is particularly prone to effects of turbulent liquid flow. All percentages differences were measured in relation to the values from the physical experiment.

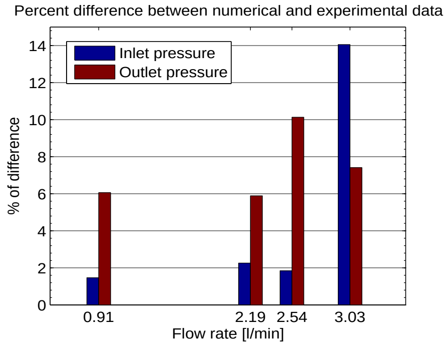

Fig. 10. The percentage difference in the results of the experiment and numerical modelling for the measured flow rates.

5. CONCLUSION

Difference of the results obtained in four numerical flow analyses in relation to a experimental data was lower than 10% for the pressure measured in front of the valve. However, for the outlet pressure it was lower than 2%. Numerical modelling results were out of the range of measurement error, they were of the same order of magnitude as experimental ones. Hence, it can be concluded that the results of numerical modelling were in compliance with the experimental data. These studies confirmed that numerical flow analysis can be used to predict and analyze the conditions and phenomena in physical systems. This may prove beneficial to the overall designing process and it would allow to improve the performance of the studied artificial heart valve.

6. ACKNOWLEDGEMENTS

Experimental and numerical analyses were performed in Foundation of Cardiac Surgery Development in Zabrze. The authors are heartily thankful to Dorota Jurkojć and Jolanta Filipek for their support and invaluable help.

BIBLIOGRAPHY

[1] ALTYNTSEV I., Flow analysis in centrifugal pump, Computer Methods in Material Science, Vol. 11, No. 1, Kraków, 2011, pp. 196–201.
[2] CHANG-HSIEN T., Numerical studies of flow field across an in vitro valve, 4th Pacific Symposium on Flow Visualization and Image Processing, Thailand, 2003.
[3] YOGANATHAN P., Flow in prosthetic heart valves: state -of-the-art and future directions, Annals of Biomedical Engineering, Vol. 33, No. 12, USA, 2005, pp. 1689-1694.