Publishing House
AKAPIT

COMPUTER METHODS IN MATERIALS SCIENCE

Informatyka w Technologii Materiałów

Vol. 11, 2011, No. 1

FLOW ANALYSIS IN CENTRIFUGAL PUMP

IEVGENII ALTYNTSEV[1,2]*, ANDRZEJ KORCZAK[2], TOMASZ SYNOWIEC[2], WOJCIECH BUJOK[1], MACIEJ DARŁAK[1], ROMAN KUSTOSZ[1]

[1] Artificial Heart Laboratory, Foundation of Cardiac Surgery Development, Zabrze, Poland
[2] Faculty of Energy And Environmental Engineering, Silesian University of Technology, Gliwice, Poland
**Corresponding author: altei@frk.pl*

Abstract

The paper describes numerical and experimental study of centrifugal pump developed to assist left ventricle of the human heart. Usage of the modern computer software provides wide possibilities of studying complicated flow processes and visualization of the obtained results.

The numerical pump model, based on theoretical study, was developed in Solid Edge. The mesh of the pump flow tract for numerical calculation was made in CFX-Mesh and ICEM CFD. The numerical calculations of the steady fluid flow in the centrifugal pump were performed to define the characteristics of the pump operating parameters at various flow rates. The construction of centrifugal pump flow tract was analyzed using Computational Fluid Dynamics (CFD) methods. Modifications of the geometry of flow tract were made by pump's constructors. Changes concerned the pump flow analyses obtained from the numerical calculations. Dangerous areas in the flow, where the level of the shear stress achieved the maximal values, were defined. The character of the fluid flow in different parts of the pump was investigated.

To perform the experimental validation of pump characteristics a laboratory stand was made. A series of measurements was performed to obtain pump performance characteristics. A comparison of the numerical calculation results and experimental measurements has shown differences between achieved results.

Key words: centrifugal pump, left ventricular assist device, numerical calculation, CFD

1. INTRODUCTION

In the past few decades the technique of insufficient human heart assistance has began to develop rapidly, especially in the area of the development of pumps and control equipment. Different types of pumps are in use in heart support systems to ensure the required blood flow in case of the heart insufficiency. All such pumps can be divided into two types: displacement pumps and rotary pumps.

Cardiac assist systems, involving displacement pumps, have been most commonly used in clinical practice in the past. The usage of rotary pumps has been increasing in recent years. These pumps have several advantages, but also a number of dangers pointed out by Reul and Akdis (2000).

Artificial Heart Laboratory (Foundation of Cardiac Surgery Development) focuses its work on improving of the clinically used pulsatile pump POLVAD and also on developing new constructions of such pumps. It is performed as a part of the national program "Polish Artificial Heart". Furthermore, the work includes development of implantable rotary blood pumps for supporting the left heart.

Numerical flow analysis of the model of centrifugal pump designed by Korczak et. al. (2010) is the object of this work. The first centrifugal blood pump was developed in 1960's. Since then, the de-

ISSN 1641-8581

velopment of new generation clinical blood pumps has been continued in the world. There have been huge changes in technology and principles of pumps design in past few decades (Nosé et al., 2000; Tang et al., 2009). Application of rotary pumps has a number of advantages but it also brings about a number of hazards which should be taken into account at the design stage. The level of modern computer technology development and software for numerical fluid flow calculations provides wide opportunities for a researcher to analyze their models at the stage of design. Numerical simulations can be useful, if not to completely replace the experiment, then, at least, to significantly reduce the number of experimental studies during the development process. Furthermore, there are wide opportunities for visualization of the processes which occur during the device's operation.

Objectives

The aim of this work is to conduct an analysis of the flow in the centrifugal pump based on the numerical calculations and comparison of numerical and experimental data.

2. MATERIALS AND METHODS

A series of numerical calculations and a laboratory experiment were performed. Subsequently, data analysis and a comparison of the obtained results were conducted in this work.

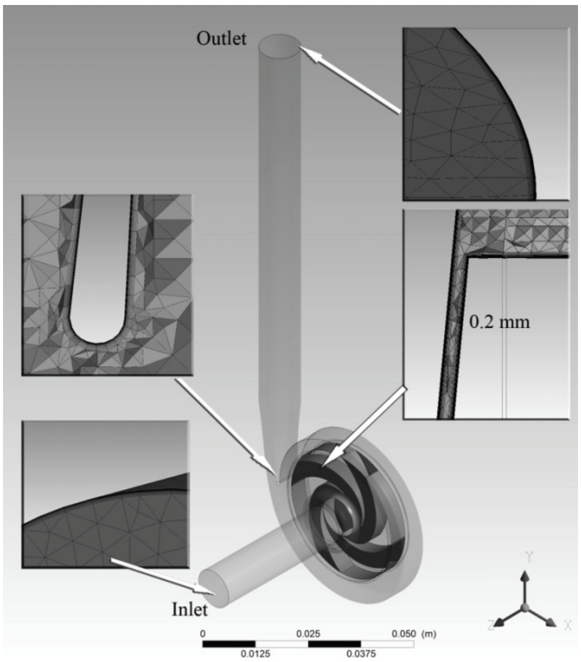

Fig. 1. Computational domain and exemplary numerical mesh in the important places.

Numerical model

ANSYS CFX *(ANSYS, Inc., USA)* software package was used to solve the problem of determining the flow parameters in the centrifugal pump. Its computational code is based on finite volume method for solving Navier-Stokes equations, describing the flow of a viscous liquid. The software also includes the most frequently used and effective models of turbulence. Hence, the investigation of the fluid flow in the channels of the centrifugal pump is the object of this study. The process of solving the equations of fluid flow numerically requires the discretization of a geometrical model. It means that it is necessary to create a high-quality mesh for numerical calculations. In this work, it was created in the following programs: CFX-Mesh and ANSYS ICEM CFD *(ANSYS, Inc., USA)*. The parameters related to the quality of the mesh have essential influence on the obtained results. Their failure could lead to results that do not correspond to reality. The mesh composed of tetrahedral elements and a wall boundary layer, was generated. The presence of the boundary layer, consisting of prismatic elements, helped to describe in detail the flow near the pump channel walls, especially in tiny gaps. In channels of 0.2 mm width, the near-wall layers of 0.07 mm were created. Each of them was divided into 15 sub-layers (figure 1) to fulfill the requirements for y+ parameter. The boundary value of the dimensionless parameter y+ for each turbulence model is different. It is necessary to create a mesh for which this parameter does not exceed that value (Song et al., 2003; Kido et al., 2006).

The numerical mesh composed of 16×10^6 elements and 6×10^6 nodes was generated for CFD calculations. That size of mesh requires a great deal of time and enormous computing power. However, that mesh enables a good description of the flow in the whole domain.

The flow part of the pump consists of rotating (impeller channels) and stationary (supply, spiral casing, discharge pipe) parts. The fluid domain was divided into the rotor and stator parts. The interaction between them was performed by means of interfaces (used Stage and Frozen rotor types). Steady-state analyses were performed to simulate the flow in the pump. The standard SST turbulence model was used in this work to solve the problem of numerical simulation of the three-dimensional flow of a viscous fluid in pump's channels of a complicated geometry. The SST turbulence model combines the advantages of k-ω and k-ε models in the boundary layers and in the middle of the channel, respectively. This model is widely used for numerical simulations of flows in blood pumps and gives good results.

To obtain the pump characteristics, calculations were performed using the Newtonian model of water (density 997 kg/m^3, dynamic viscosity 8.899×10^{-4} Pa·s). The total pressure at the inlet was set 98100 Pa, while reference pressure in the whole fluid domain was equal to 0 Pa. The mass flow boundary condition was set at the outlet of the pump. It was calculated from a given flow rate (from 0.05 dm^3/s (3 L/min) to 0.15 dm^3/s (9 L/min), details shown in figure 9). The flow rate was varied to obtain the pump characteristics. Domain walls were smooth and the flow velocity on them was equal to 0 m/s.

Experimental model

To test the pump model in the laboratory an experimental stand was prepared. The flow rate through the pump was measured with the ultrasonic flowmeter FLUXUS® ADM (*FLEXIM, USA*). The accuracy of volume measurements equals ±1÷3% of the measured value depending on an application.

Figure 2 shows a schema of the experimental stand. To define the parameters of the pump, pressure measurements were performed at the inlet and outlet of the pump, as well as at the diffuser inlet, which made it possible to obtain a more complete picture of the flow. The accuracy of pressure sensor measurements is 0.25% in the pressure range of -100÷100 kPa. The special physical model of the impeller was created for laboratory tests. Between the pump's shaft and the pump housing the sliding seal was applied to reduce the leakage. The use of this type of seal complicates the task of the construction evaluation, due to an increase in the energy loss in the seal. The physical evaluation experiment should be treated as preliminary, due to imperfections in the physical configuration.

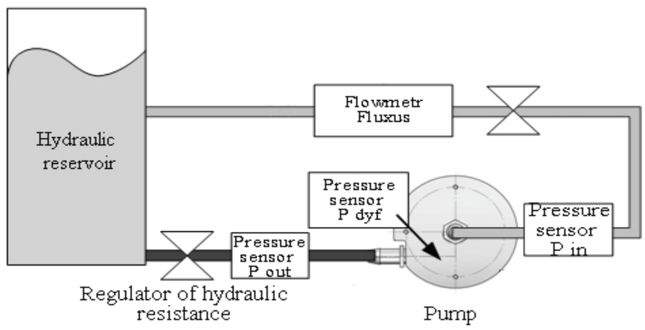

Fig. 2. *Scheme of experimental stand.*

Series of measurements for various flow parameters and different speeds of the impeller rotation were carried out in the laboratory tests. As a result, the pump performance characteristics for different conditions of its operation were obtained.

3. RESULTS

Calculations of the flow in the centrifugal pump were performed to determine the pump characteristics and to visualize the processes in the flow part.

Figure 3 shows vector fields in the longitudinal and cross section planes of the pump. The vectors show that the maximum fluid velocity appears at the maximum radius of the impeller. A comparison of the results of calculations under various boundary conditions, it could be observed that the zone formation of a vortex at the inlet of the diffuser increased with the increase of the flow through the pump. Thus, at 0.06 dm^3/s (3.6 L/min) flow rate condition, the fluid enters the diffuser smoothly and the vortex does not appear as shown in figures 4a, 5a. A small vortex starts forming, while the outlet flow rate is 0.09 dm^3/s (5.4 L/min), which can be seen in figures 4b, 5b. Flow rate value of over 0.09 dm^3/s (5.4 L/min), vortices could be observed in figures 4c, 5c.

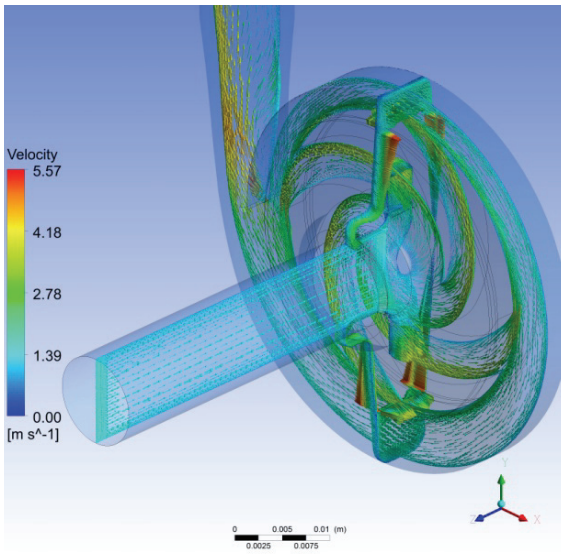

Fig. 3. *The fields of velocity vectors in the longitudinal and cross-sectional planes in the flow part at flow rate Q = 0.13 dm^3/s (7.8 L/min).*

Figure 5 shows fluid streamlines in the pump at different flow rate values. The vortices formed in the diffuser (figure 5 b, c) are sufficiently significant. It means that great losses of energy are present in that place. A sharp decrease of velocity exists in the vortex region. It can cause the possibility of blood clots formation.

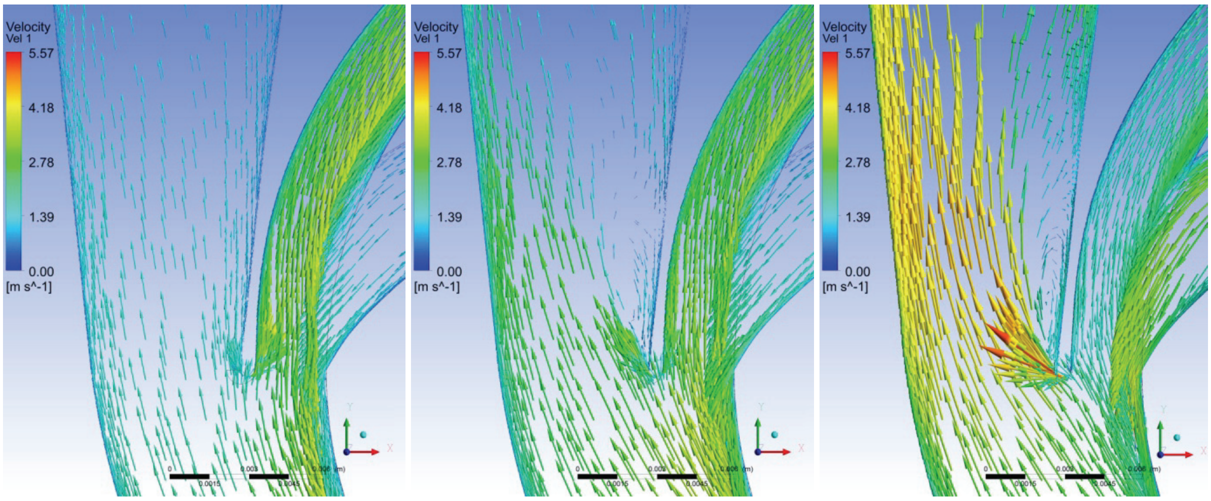

__Fig. 4.__ Fields of vectors in the diffuser at different flow rates: a) Q = 0.06 dm^3/s (3.6 L/min), b) Q = 0.09 dm^3/s (5.4 L/min), c) Q = 0.13 dm^3/s (7.8 L/min).

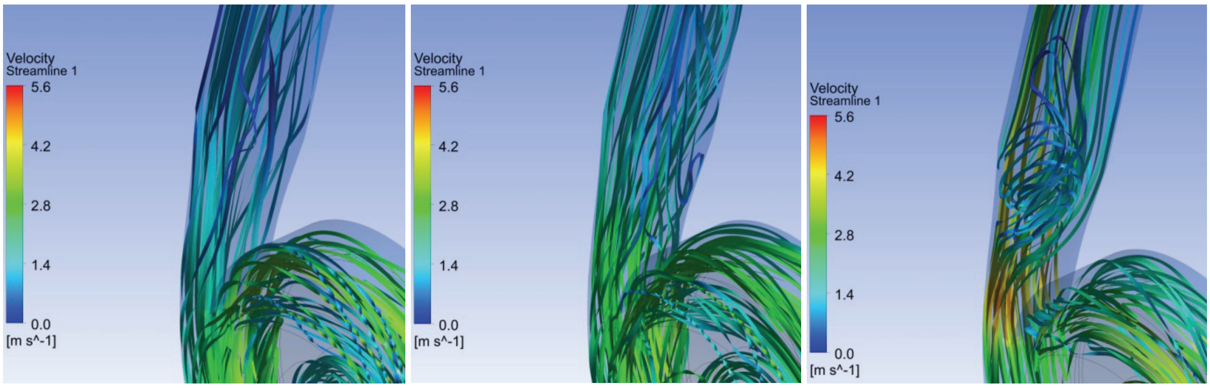

__Fig. 5.__ Streamlines in the centrifugal pump at the flow rate values: a) Q = 0.06 dm^3/s (3.6 L/min), b) Q = 0.09 dm^3/s (5.4 L/min), c) Q = 0.13 dm^3/s (7.8 L/min).

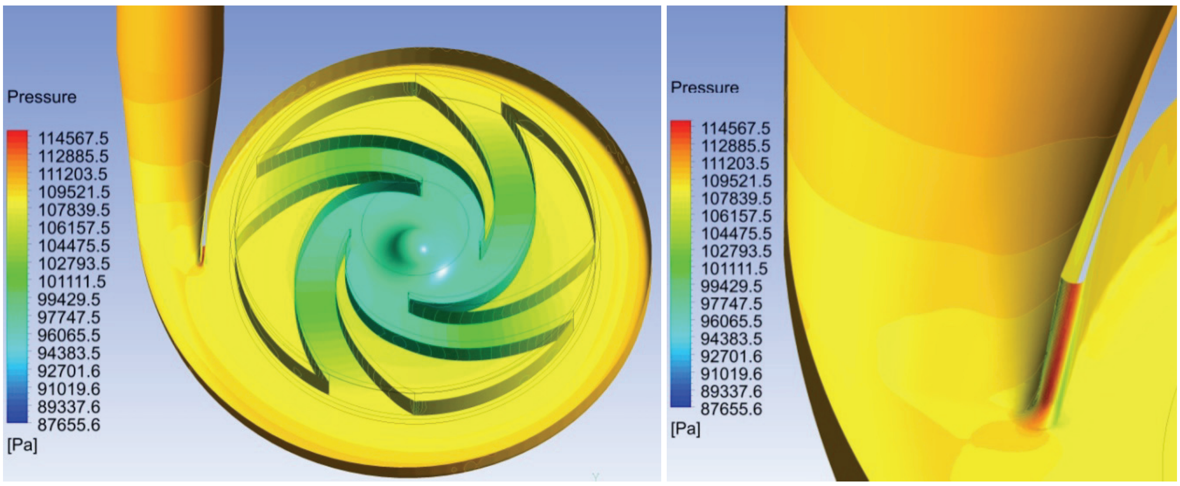

__Fig. 6.__ The pressure field at the pump walls at flow rate Q= 0.09 dm^3/s (5.4 L/min).

As can be seen in figures 4c and 5c, high velocities of fluid particles are observed at the inlet of the diffuser at the flow rate of 0.13 dm^3/s (7.8 L/min).

Figure 6 shows pressure distribution on the impeller and spiral casing walls. It is necessary to obtain the pump operating conditions, which will exclude the blood hemolysis. Such a phenomenon is influenced by the shear stress and residence time of blood cells under such loads. Greatest change in velocity and pressure is observed In the cut-off of the diffuser. It affects the magnitude of the shear stress. Figure 7 shows the field of shear stresses on the walls of the pump flow part channels.

High stresses occur on the blades' surfaces and at the cut-off of the diffuser. However, time of an influence of high values of shear stresses on blood cells has to be taken into account and it is necessary to investigate it in future.

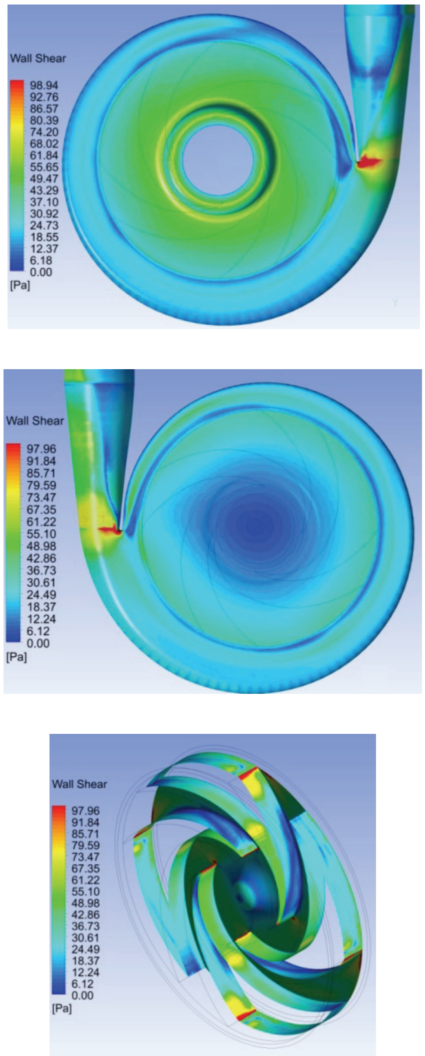

Fig. 7. *The shear stress on the walls of the pump at flow rate Q = 0.13 dm³/s (7.8 L/min).*

The highest values of shear stresses were observed at the level of 580 Pa. Areas of high shear stresses which are shown in figure 7 are much more smaller than showed by White (2008). Figure 8 shows exemplary results of time of fluid particles route in the area of the diffuser inlet. Presented time on streamline (figure 8) cannot be treated as the exposure time of blood cells, but only as an approximate evaluation of the fluid structure in this region.

To validate numerical calculations the experimental test was made, hence the laboratory and numerical characteristics comparison was illustrated in figure 9. The chart presents the chosen numerical and experimental pump characteristics for the variety of impeller rotational speeds. The pump model roughness, the indefinite energy loss in the seal, the impeller position precision in the pump housing (possible friction) should be treated as the research imperfection, and due to these the experiment is going to be repeated in the near future. However, at this moment, the preliminary CFD and experimental model comparison was done. Qualitatively the characteristics are similar, but there is a difference between pressure values (shown in figure 9). The maximum difference between the results achieves 28%.

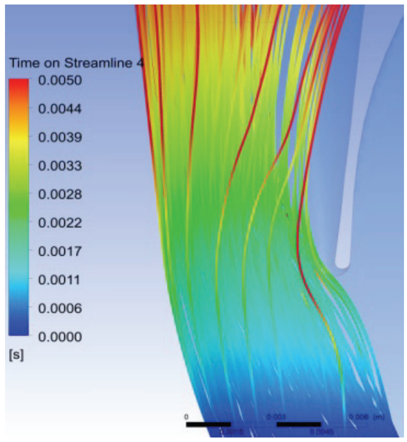

Fig. 8. *Time on streamline at the inlet of the diffuser at flow rate Q = 0.13 dm³/s (7.8 L/min).*

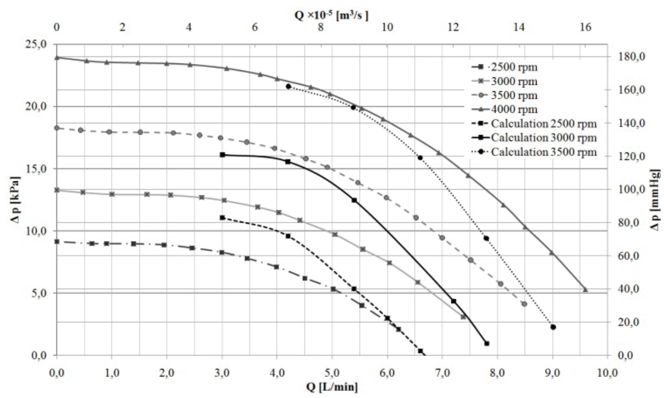

Fig. 9. *Characteristics of the centrifugal pump for different rotational speeds of the impeller.*

4. DISCUSSION

The vortices in the diffuser, which could be observed in figures 4 and 5, are phenomenon, which are present in all centrifugal pumps. Further changes in the geometry of the spiral casing will probably reduce vortices in the diffuser and the maximum value of the shear stress on walls of the pump flow part.

A high level of shear stress in some areas of the pump flow part has a negative meaning, additionally it is necessary to investigate the exposure time of blood cells, to define precisely the level of hemolysis.

The difference obtained from experimental and numerical data results is at a high level. It could be the reason of several factors like: the pump model roughness, the indefinite energy loss in the seal, the impeller position precision in the pump housing, and it will be investigated redefined in the future.

5. CONCLUSIONS

Numerical calculations, using ANSYS CFX, allow one to obtain characteristics of the pressure, velocity and shear stress distribution. The comparison of the flow in the diffuser at different conditions, shows that increasing flow rate through the pump induces the growth of the area of vortices. It means that energy losses in that area are growing also. Moreover, the sharp drop of velocity in the middle of the vortex could be the reason of possibility of the formation of blood clots.

Comparing results of numerical calculations and experimental studies large differences can be seen. This requires further studies to determine reasons of the differences in the results.

REFERENCE

Kido, K., Hoshi, H., Watanabe, N., Kataoka, H., Ochuchi, K, Asama, J., Shinishi, T., Yoshikawa, M., Takatani, S., 2006, Computational Fluid Dynamics Analysis of the Pediatric Tiny Centrifugal Blood Pump (Tiny Pump), *Artif Organs*, 30 (5), 392-399.

Korczak, A., Kustosz, R., Bujok, W. Synowiec, T. Altyntsev, E., Pompa odśrodkowa jednostrumieniowa. Zgłoszenie patentowe nr P.390781 zgłoszono 2010-03-22 (in Polish)

Nosé, Y., Yokishawa, M., Murabayashi, S., Takano, T., 2000, Development of Rotary blood Pump technology: Past, Present, Future, *Artif Organs*, 24(6), 412-420.

Reul, H. M. and Akdis, M., 2000, Blood pumps for circulatory support, *Perfusion*, 15, 295-311.

Song, X., Wood, H. G., Day, S. W., Olsen, D. B., 2003, Studies of Turbulence Models in a Computational Fluid Dynamics Model of a Blood Pump, *Artif Organs*, 27 (10), 935-937.

Tang, D.G., Oyer, P.E., Mallidi, H.R., 2009, Ventricular Assist Devices: History, Patient Selection, and Timing of Therapy, *J of Cardiovasc Trans Res*, 2, 159-167.

White, D., 2008, CFD Analysis of the HeartWare® VAS Blood Pump, *Proc. Conf. International ANSYS Conference: Inspiring Engineering*, Pittsburgh, Pennsylvania, USA.

ANALIZA PRZEPLYWU W POMPIE ODŚRODKOWEJ

Streszczenie

W artykule przedstawiono numeryczne i eksperymentalne badania pompy odśrodkowej do wspomagania lewej komory serca człowieka. Korzystanie z nowoczesnych programów komputerowych oferuje szerokie możliwości w zakresie badań skomplikowanego procesu przepływu i wizualizacji uzyskanych wyników.

Model numeryczny pompy, zbudowany na podstawie badań teoretycznych, został skonstruowany w programie Solid Edge. Siatka części przepływowej pompy dla obliczeń numerycznych została zbudowana w programach CFX-Mesh i ICEM CFD. Przeanalizowano konstrukcję pompy odśrodkowej wykorzystując metody obliczeniowej dynamiki płynów (CFD). Przeprowadzono numeryczne obliczenia ustalonego przepływu w pompie odśrodkowej, celem określenia charakterystyki parametrów pracy przy różnych parametrach przepływu. Zostały wykonane zmiany geometrii kanału przepływowego przez konstruktorów. Są one oparte na wynikach analizy przepływu w pompie, uzyskanych przy pomocy obliczeń numerycznych. Zdefiniowano niebezpieczne strefy w strumieniu, gdzie poziom naprężeń ścinających osiąga maksymalne wartości. Zbadano charakter przepływu cieczy w różnych elementach pompy.

Celem otrzymania eksperymentalnych charakterystyk pompy, stworzono stanowisko laboratoryjne, na którym przeprowadzono serie pomiarów. Porównanie rezultatów obliczeń numerycznych i pomiarów eksperymentalnych pokazuje, że istnieje różnica między wynikami.

Received: November 10, 2010
Received in a revised form: November 26, 2010
Accepted: December 15, 2010