Archives of Materials Science and Engineering

Volume 64
Issue 2
December 2013
Pages 198-204

International Scientific Journal
published monthly by the
World Academy of Materials
and Manufacturing Engineering

Finite Element Method application for modelling of internal oesophageal prosthesis

A.J. Nowak [a,*], L.A. Dobrzański [a] , R. Rybczyński [b], R. Mech [b]
[a] Institute of Engineering Materials and Biomaterials, Silesian University of Technology,
ul. Konarskiego 18a, 44-100 Gliwice, Poland
[b] Institute of Materials Science and Applied Mechanics, Faculty of Mechanical Engineering,
Wroclaw University of Technology, ul. Smoluchowskiego 25, 50-370 Wrocław, Poland
* Corresponding e-mail address:agnieszka.j.nowak@polsl.pl

Received 06.10.2013; published in revised form 01.12.2013

ABSTRACT

Purpose: The general topic of this paper is the computer simulation with the use of finite element method for determining the internal stresses in the internal oesophageal prosthesis based on long-fibre composite material.

Design/methodology/approach: Modelling of stresses in the internal oesophageal prosthesis was performed with the help of finite element method in ANSYS environment. Application of Finite Element Method was discussed and essential advantages resulting from application of it are pointed

Findings: The presented model meets the initial criteria, which gives ground to the assumption about its usability for determining the stresses in the internal oesophageal prosthesis, employing the finite element method using the ANSYS program. The computer simulation results correlate with the experimental results.

Research limitations/implications: Applied Finite Element Method enables modelling of stresses and deformations arises in composite material in the conditions representing real experimental investigations as well as condition similar to those prevailing in human body after prosthesis implementation.

Originality/value: Developed simulation simplifies and reduces cost of optimization of the internal oesophageal prosthesis properties by simulation of this properties without necessity of additional laboratory investigations.

Keywords: Finite Element Method; Materials; Composites; Engineering polymers; Biomaterials; Oesophageal prosthesis

Reference to this paper should be given in the following way:

A.J. Nowak, L.A. Dobrzański , R. Rybczyński, R. Mech, TFinite Element Method application for modelling of internal oesophageal prosthesis, Archives of Materials Science and Engineering 64/2 (2013) 198-204.

METHODOLOGY OF RESEARCH, ANALYSIS AND MODELLING

1. Introduction

Nowadays in Poland, depending on the level of cancer development, there are used interchangeably, stents (in order to expand a contracted oesophagus), natural „prostheses" (vital tissues of the patient - the remnants of digestive system, e.g. stomach - are used for reconstruction of the loss) and so called extracorporeal tube (used in the case of loss of large part of digestive system like resection of oesophagus, stomach and duodenum). The last case, extracorporeal tube, is connected with several health complications as well as limited mobility of the patient, huge discomfort and social exclusion [6-9, 20]

A lack of existing true solution of nutrient transport of patients with removed large parts of the digestive system justifies the need of searching for the solution leading to development of internal oesophageal prosthesis. The prosthesis could be successfully used in digestive surgery, especially in the case of patients with highly developed pathological changes of this organ [6,9,20].

Because there is no possibility of determination of mentioned prosthesis after implantation in "in vivo" conditions, it was decided to make numerical analysis of the prosthesis with the use of Finite Element Method. The literature studies shows that that kind of analyses have not been done yet [5-7, 20-30]

The main assumption of Finite Element Method is discretization of continuous geometrical systems. It is based on division of main system onto finite number of subregions. Complicated parts, and even very complex units are represented by simple components. The more such components, the more precise calculations are. In construction mechanics this method is used in almost every kind of problems. The designing of own computer programs based on Finite Element Method requires very good grounds in the range of mathematics, numerical methods and theory of analysed problems. [1,4, 10-20, 23-29].Whereas applying ready-made, checked packages of FEM, there is only necessity of understanding of the method and its mathematical basics and knowledge of used program and knowledge of simulated phenomena. In every case, solutions calculated in FEM program should be verified by simplified analytical calculations and control of quality correctness of given partial results (e.g. continuity of stress field or compliance with boundary conditions) [3,5,7,9,13,19,30].

In many composite constructions, especially multilayer ones, there can be distinguished layers with orthotropic properties, i.e. where material properties changes in perpendicular directions other than main axes of established coordinate system. This is in the case of considering the construction made from resins reinforced with continuous fibres as well as microstructure of some ceramic or metallic materials [17, 18, 20, 22, 30].

Due to specific functional properties of internal oesophageal prosthesis, it has been decided to evaluate the composite material which is characterized by biocompatibility and high purity as well as specified mechanical properties. Taking into consideration the length of the prosthesis and high probability of occurrence of internal tissues injures resulting from prosthesis presence it has been specially emphasized the adequate elasticity and biocompatibility of the material to avoid such type of medical complications [5-9, 20].

After initial verification, following components of composite material have been chosen:

- Aramid fibre characterized by following properties:
  - Density 1.44 g/cm^3;
  - Young's modulus 105 GPa;
  - Tensile strength 3053 MPa;
  - Ultimation at rapture 2.70%;
  - Decomposition temperature 490°C;
  - Contraction in hot air (15 min at the temperature of 190°C) => 0.1%;
  - Thermal resistance (48h at the temperature of 200°C) => 90%.
- Medical silicon, with following physical properties:
  - Boiling temperature/ranges > 82°C;
  - Flash-point 13.3°C (Pensky-Martens closed crucible);
  - Specific gravity 0.865 g//cm^3;
  - Viscosity 132 cSt (at the temperature of 25°C).

In the presented article, there is considered problem of modelling multi-layered composite materials, there are presented basic mechanical relations of such materials.

2. Investigation methodology

Considered in the work orthotropic material contain nine independent coefficients in the matrix determining relation between components of stress and strain states. These coefficients interpretational reflects materials constants, i.e. Young's modulus E (E11, E22, E33), constant Poisson ν (ν21, ν31, ν32) and Kirchoff's modulus G (G23, G31, G12). Modulus E11, E22, E33 are determined from static tensile test along main orthotropic axes, modulus G23, G31, G12 requires appropriate shearing test, and each constant νij is determined by ratio - εjj/ εii for the specimen loaded only by tensile stress σii [2,17,18, 21,22,30]. The assumptions made in this work are as follows:

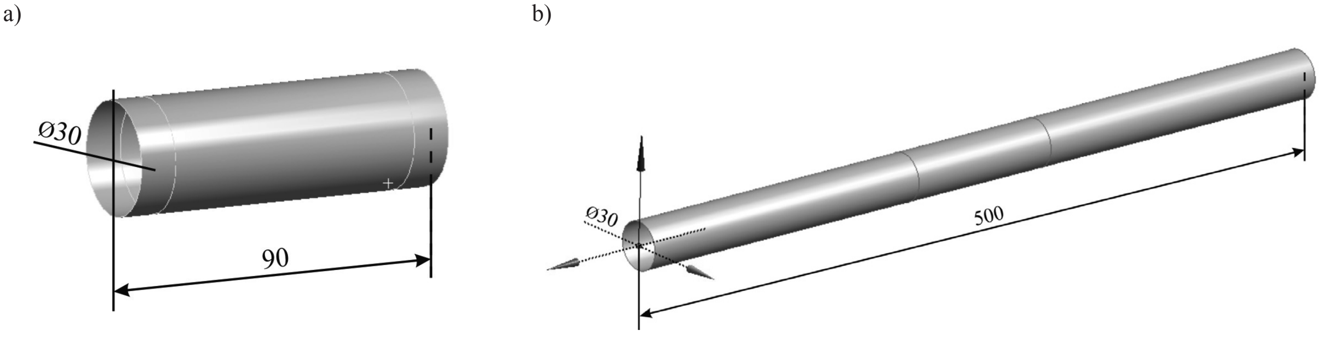

Fig. 1. Geometrical model of internal oesophageal prosthesis: a) the model of type 1;b) the model of type

- modules E11 and E22 are equal and have been determined from static tensile test of newly evaluated composite material with fibres arranged in the angle of 0° in the matrix; additionally from this test $\nu21$ has been determined on the basis of measurements of strains in longitude and across direction;
- parameter E33, that is stiffness in direction perpendicular to the layer of composite material, as well as $\nu31 = \nu32$ and G23 = G31 have been determined from tensile test of pure medical silicon - matrix treated as an isotropic material,
- parameter G12 has been determined on the basis of results of static tensile test of composite material with the fibres arranged in the angle of 45°.

Numerical analysis has been made in ANSYS Workbench 12.0 program [30].

Because there is no possibility of determination of mentioned prosthesis after implantation “in vivo” conditions, additional model representing done pressure tests was made. This model enables verification of constructional and material assumptions correctness of the full model of the prosthesis.

In the work, there are presented geometrical models on the Figs. 1a and 1b. Discretization of these models was made using multi-layered elements of the SHELL 181 type.

There has been selected covering element because of not large thickness of the object in relation to its real dimensions. This element models in which bend stresses dominates very well, even though it does not take into consideration stresses perpendicular to the surface. The finite element has four nodes and the thickness is the parameter affecting its stiffness (it is not geometrically modelled). In the first case model was composed of 3843 elements and 3906 nodes, while in the second one of 5536 elements and 5568 nodes. The thickness of finite covering elements is estimated as an average of real dimensions of the objects. The established coordinate system takes into account directions of orthotropy occurred in real material (Fig. 2).

Determined boundary conditions represent supports and loads occurring in considered real objects:

- pressure exerted on internal surface of the prosthesis which simulates experimental test in laboratory condition,
- load by forces simulating internal organs pressure when the prosthesis is implanted into the patient.

Loads and strengthens of both models are presented on Figs. 3 and 4.

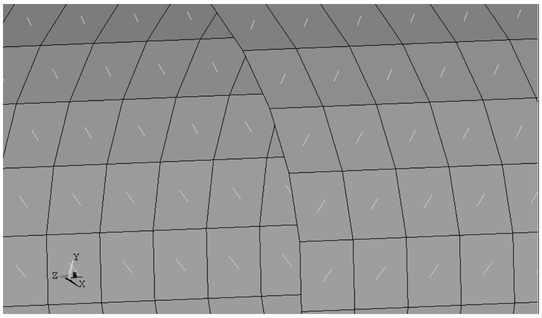

Fig. 2. The setting of the slope of orthotropic layers in an ANSYS Classic

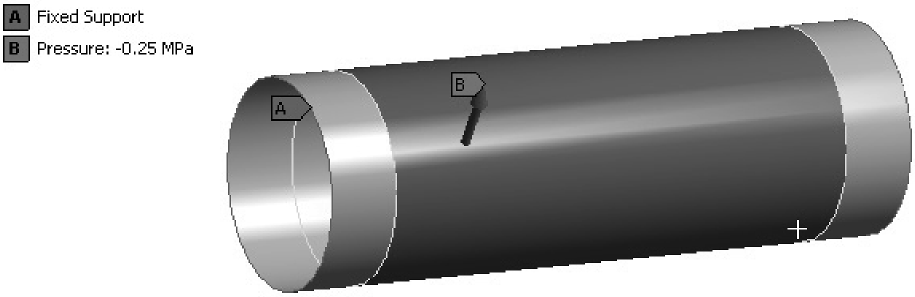

Fig. 3. The adopted boundary conditions of the model- type 1

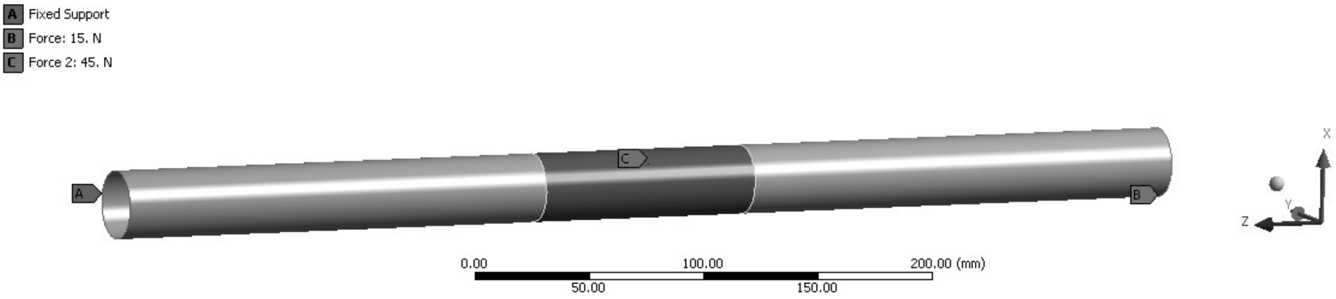

Fig. 4. The adopted boundary conditions of the model - type 2

Forces for the second geometrical model has been chosen basing on virtual experiment to obtain displacements similar to real one in human body after implantation of the prosthesis according to the considerate third dimension of the prosthesis. (Fig 5). Results of calculation analysis made using Finite Elements Method enables to determine if in the option of most loaded prosthesis implantation its diameter would be closed or destroyed.

According to Huber-Misesa-Henck's hypothesis obtained results of displacement and stresses analyses are reduced values.

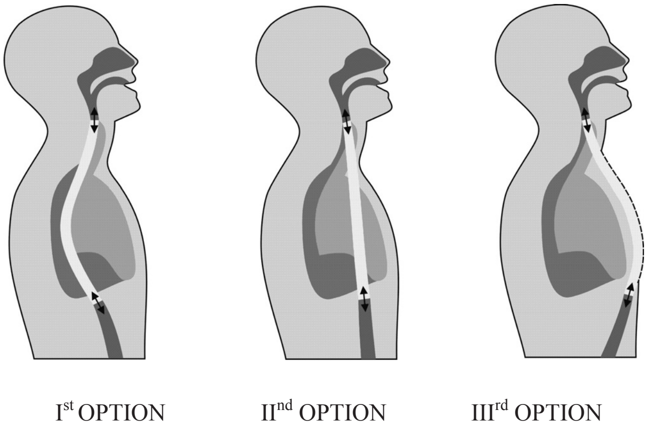

Fig. 5. Adopted options for a possible oesophageal prosthesis implantation intra-corporeal.

3. Investigations results

Symbols of prostheses prototypes assumed in this work are listed in Table 1, and in the Fig. 6 there are presented exemplary results of numerical analysis based on Finite Elements Method collected as stress pattern maps in composite material dedicated for oesophageal prosthesis.

Evaluated model of prosthesis enables simulation of loads in the most difficult third prosthesis implantation option.

It has been checked if the diameter of the prosthesis would stay open or will be closed and damaged.

In order to verify applied numerical analysis of the prosthesis laboratory strength tests of the tube specimens have been used. Determined characteristics of tubular specimens applying Finite Elements method are consistent with results obtained experimentally (Fig. 7).

On the basis of performed numerical tests it can be emphasized that reduced stresses for prostheses with the beam roll „19\2_1" equals to 15,33 Mpa, and for the prosthesis with the beam „23\2_1"and „43_1" respectively equals to 11.07 and 11,05 MPa. This difference results from orthotropic properties of the composite material. Prostheses with the wrap beaml „23\2" and „43\2" are characterized by small, nearly one percent difference in simulated results depending on the number of layers. In the prosthesis with the beam roll 19/2 this difference reaches 31%.

Table 1.
Symbols adopted for prototype prosthesis

The number of fibres bundles	19			23			43		
The type of roll	2			2			2		
The number of the layers	2	3	4	2	3	4	1	1.5	2
Designations adopted	19\2_1	19\2_2	19\2_3	23\2_1	23\2_2	23\2_3	43\2_1	43\2_2	43\2_3

The smallest strain have been found for the prosthesis with the beam roll „43\2_3" (22.92 mm) and „23\2_3" (23.60 mm) characterized by the highest stiffness also in experimental tests. For the prosthesis with the beam roll 19/2 the highest movements were detected in the range 25.86-44.81 mm as regards other prosthesis.

Performed numerical analyses applying Finite Elements Method confirms real properties of oesophageal prostheses made from newly-evaluated composite material. It is implied by the fact of high resistance to closing prosthesis diameter during or after implantation. Obtained results of stresses for each considered type of prosthesis suggest that there is no possibility of any damage of the prosthesis during its exploitation.

4. Conclusions

Worked out numerical analysis using Finite Elements Method in the environment of the ANSYS Workbench 12.0 program for complex properties of internal prosthesis of oesophagus allows to determine state of displacements, strains and stress in two load options according to Hubera-Misesa-Henck's effort hypothesis. This investigations have been made in order to determine if considered oesophageal prosthesis will deform in implantation conditions leading to closing its diameter what could be very dangerous for patients health and even life and resulting in the necessity of reoperation of already weakened organism. In the analysis, there has been assumed mechanical properties of composite material, diversified in the type of warp beam and the layers, determined experimentally. The evaluated simulation enables prediction of material properties, and thus oesophageal prosthesis with higher or lower number of layers in given type of warp beam. It is possible to support optimization of the works related with the selection of appropriate prosthesis stiffness considering its future application and required comfort of its exploitation.

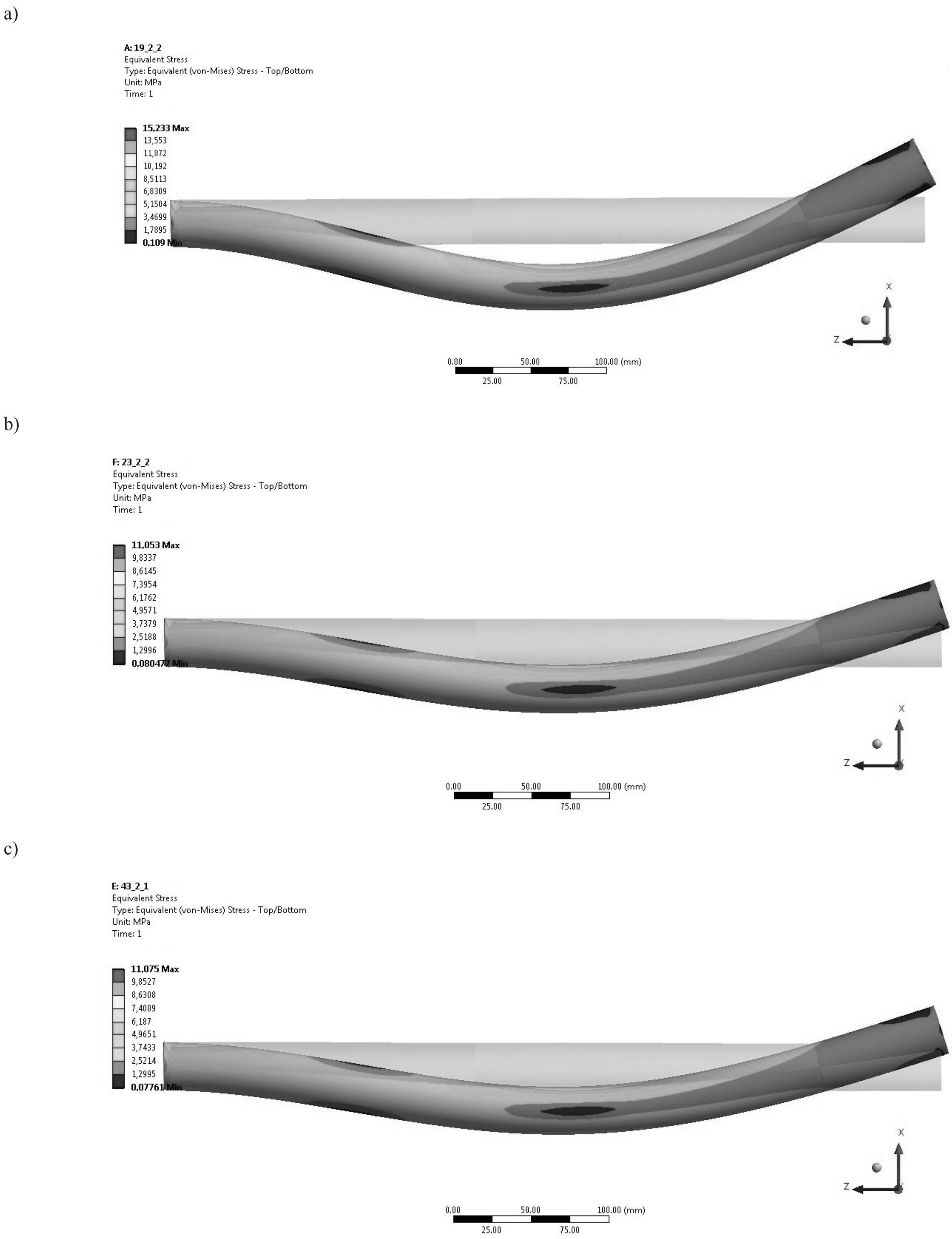

Fig. 6. The adopted boundary conditions of the model: a) 19/2_1; b) 23/2_1; c) 43/2_1

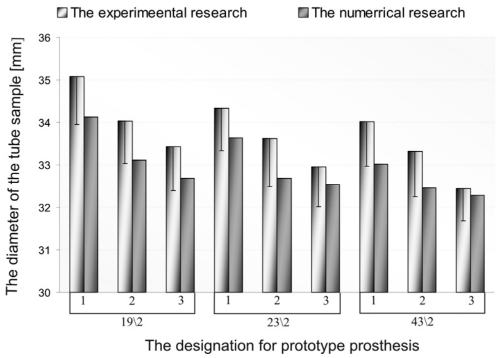

Fig. 7. Comparison of experimental results and numerical changes in diameter of pipe samples

Acknowledgements

Agnieszka J. Nowak is a holder of scholarship from project POKL.04.01.01-00-003/09-00 entitled "Opening and development of engineering and PhD studies in the field of nanotechnology and materials science" INFONANO, co-founded by the European Union from financial resources of European Social Fund and headed by Prof. L.A. Dobrzański.

HUMAN CAPITAL
NATIONAL COHESION STRATEGY

PROJEKT infonano

EUROPEAN UNION
EUROPEAN
SOCIAL FUND

References

[1] T. Antosik, J. Awrejcewicz, Numerical and experimental analysis of biomechanics of three lumbar vertebrae, Journal of Theoretical and Applied Mechanics 3 (1999) 413-434.

[2] K.J. Bathe, Finite element procedures in engineering analysis, Publication of Hall, New Jersey, 1982.

[3] P. Borkowski, K. Kędzior, G. Krzesiński, K.R. Skalski, P. Wymysłowski, T. Zagrajek, Numerical investigation of a new type of artificial lumbar disc, Journal of Theoretical and Applied Mechanics 42/1(2004) 253-268.

[4] D. Chua, Finite elements simulation of stent expansion, Journal of Materials Processing Technology 120 (2002) 335-340.

[5] L.A. Dobrzański, A. Pusz A, A.J. Nowak, The effect of micropores on output properties of laminates materials with assumed medical implantation, Journal of Achievements In Materiale and Manufacturing Engineering 37/2 (2009) 408-415.

[6] L.A. Dobrzański, A. Pusz, A.J. Nowak, M. Górniak, Constructional model of internal oesophageal prosthesis, Archives of Materials Science and Engineering 42/2 (2010) 69-76 2010.

[7] L.A. Dobrzański, A. Pusz, A.J. Nowak, M. Górniak, Application of computer techniques in modelling of biofunctional implants, Journal of Achievements in Material and Manufacturing Engineering (in print).

[8] L.A. Dobrzański, A. Pusz, A.J. Nowak, M. Górniak, Non-standard test methods for long-fibrous reinforced composite materials, Archives of Materials Science and Engineering 47/1 (2011) 5-10.

[9] L.A. Dobrzański, A. Pusz, A.J. Nowak, M. Górniak, The concept of preparation of oesophageal prosthesis based on long-fiber composite material, Journal of Achievements in Materials and Manufacturing Engineering 46/1 (2011) 18-24.

[10] A. Grabarski, I. Wróbel, Introduction to FEM, Publishing house Warsaw University of Technology, Warsaw 2008 (in Polish).

[11] W. Grzesikiewicz, A. Zbiciak, Study of generalized Prandtl rheological model for constitutive description of elastoplastic properties of materials, Journal of Achievements in Materials and Manufacturing Engineering 55/2 (2012) 501-510.

[12] A. Kocańda, O. Czyżewski, Numerical analysis of abrasive wear of forging dies, Numiform 104-106 (2001).

[13] M. Kopernik, J. Nowak, Physical and numerical modeling of functional trileaflet aortic valves, Mechanic 11 (2008) 958-963.

[14] K. Król, Finite Element Method in the design calculations, Publishing house Radom University of Technology, Radom, 2007 (in Polish).

[15] W. Kwaśny, R. Dziwis, Application of the Finite Element Method for computer simulation of aluminum stamping process, Journal of Achievements in Materials and Manufacturing Engineering 55/2 (2012) 551-555.

[16] M.R. Labrosse, C.J. Beller, F. Robicsek, M.J. Thubrikar, Geometric modeling of functional trileaflet aortic valves. Development and clinical applications, Journal of Biomechanics 39 (2006) 2665-2672.

[17] S. Łączka, Introduction to the ANSYS Finite element system, Cracow Technical University Press, 1999.

[18] T. Łodygowski, Finite element method in selected problems of mechanics end construction engineering, Publishing house University of Technology, Poznan, 1991.

[19] F. Migliavacca, Mechanical behavior of coronery stents investigated through the fine element method, Journal of Biomechanics 35 (2002) 803-811.

[20] A.J. Nowak, Structure and properties of newly developed composite material for internal prosthesis of oesophagus, PhD Thesis, Central Library of The Silesian University of Technology, Gliwice, 2011.

[21] T. Pedersen, Numerical modelling of cyclic plasticity and fatigue damage in cold-forging tools, International. Journal of Mechanical Sciences 42 (2000) 799-818.

[22] G. Rakowski, Z. Kacprzyk, FEM in mechanical design, Publication of Warsaw University of Technology, Warsaw, 2005 (in Polish).

[23] A. Śliwa, M. Matula, L.A. Dobrzański, Finite Element Method application for determining feedstock distribution during powder injection moulding, Journal of Achievements in Materials and Manufacturing Engineering 37/2 (2009) 584-591.

[24] A. Śliwa, J. Mikuła, K. Gołombek, L.A. Dobrzański, FEM modelling of internal stresses in PVD coated FGM,, Journal of Achievements in Materials and Manufacturing Engineering 36/1 (2009) 71-78.

[25] J. Stadnicki, Z. Tokarz, Analysis of deflection of composite aircraft wing, Bulletin of the Military Technical Academy 57/2 (2008) 45-54 2008.

[26] T. Stolarski, Y. Nakasone, S. Yoshimoto, Engineering analysis with ansys software, Publishing house Elsevier, United Kingdom, 2008.

[27] M. Szymiczek, G. Wróbel, M. Rojek, T. Czapla, Simulation diagnostics of the polyester-glass pipes degradation process;experimental basis, Journal of Achievements in Materials and Manufacturing Engineering 59/1 (2013) 37-47.

[28] F.D. Whitcher, Symulation of in vivo loadin conditions of nitinol waskular stent structures, Computers and Structures 64 (1997) 1007-1011.

[29] G. Wróbel, M. Rojek, M. Szymiczek, Simulation studies of fatigue degradation process with reference to composite pipes, Journal of Achievements in Materials and Manufacturing Engineering 55/2 (2012) 596-599.

[30] www.ansys.com