Journal of Achievements in Materials and Manufacturing Engineering
VOLUME 42
ISSUES 1-2
September-October
2010

Application of FEM for solving various issues in material engineering

L.A. Dobrzański, A. Pusz, A.J. Nowak*, M. Górniak
Institute of Engineering Materials and Biomaterials, Silesian University of Technology, ul. Konarskiego 18a, 44-100 Gliwice, Poland
* Corresponding author: E-mail address: agnieszka.j.nowak@polsl.pl

Received 10.06.2010; published in revised form 01.09.2010

Analysis and modelling

ABSTRACT

Purpose: The aim of this work is to present selected problems concerning the application of Finite Element Method in materials engineering on the example of chosen program which makes the most of this method to simulation.
Design/methodology/approach: Application of Finite Element Method was discussed and essential advantages resulting from application of it are pointed.
Findings: Description of the importance and the utility of FEM during solving of problems dealing with very complicated geometry complex state of loadings, various boundary conditions and/or various materials.
Research limitations/implications: The method must be applied very carefully because its results do not refer to real system but only to the model one. The obtained results of FEM calculations can be used to solve many problems at the early step of designing with success.
Originality/value: The application of FEM method during working out the internal prosthesis of oesophagus which will enable help people suffering from oesophageal cancer.
Keywords: Computational materials science; Finite Element Method; Materials; Composites; Engineering polymers; Biomaterials

Reference to this paper should be given in the following way:
L.A. Dobrzański, A. Pusz, A.J. Nowak, M. Górniak, Application of FEM for solving various issues in material engineering, Journal of Achievements in Materials and Manufacturing Engineering 42/1-2 (2010) 134-141.

1. Introduction

The Finite Elements Method (FEM), that is helpful in solving wide variety of engineering problems, was developed 40 years ago. During first fifteen years of its existence, this method had been the subject-matter of intensive studies and as a result its general rules and mathematical basis were formed. In the middle of seventies the method has already been widely used in engineering design and its more intense development has been obstructed by insufficient calculation power of computers. In the middle of eighteens, only after personal computers appeared, on which programs of large calculation units were replaced, as effective tool in solving problems, analyses and designing as FEM become widely available.

Emphasizing huge possibilities and usability of FEM, special attention shall be paid to necessity of cautious and thought-out application of that method. It must be underline that it is approximation method. Solutions that are results of this analysis are considering not the real constructional system, but their models. The difference between a real problem and the solution of the model calculations is presented on Fig. 1. There are shown multistage steps and simplifications made from the reality to the final solution. In the light of such defined process, the necessity of though-out application of MES seems to be obvious, and in particular, critical afford to analysis of obtained solutions [1-10,13,22,28].

Intensive progress of computer techniques caused that nowadays the method is one of the most important among methods of numerical analysis of a construction. The whole idea of the method is based on building complicated objects from the simple ones or division of those models into small, defined elements. From the engineering point of view FEM is a sequence of defined operations made by the engineer – the designer and the computer, during searching of solution leading from formulation of the problem to the final interpretation of results of calculations [1-5,16-23,29].

The number of equations is proportional to the number of nodes and degrees of freedom existing in a given element or area. Finite Element Method, as it has been already defined before, is a numerical technique for verifying differential equations for matrix operations, and the higher calculation power of computer the wider area of applications [9-13,16,22-25].

2. Main ideas of FEM

FEM is a numerical method used for solving engineering problems and as an approximation method is applied in pure mathematical problems in physical science. Concerning the problems when the geometry of the construction is very complicated, including various boundary conditions, and constructions are made from different materials, so in the case of problems in which it is usually impossible to obtain analytical solution, it is the most useful method [4,5,12-18].

It is used discretization of continuous system in Finite Element Method. The matter consists in division of continuous (uniform) system on equivalent system (called discrete model) of smaller bodies or elements (called finite elements). Those elements are connected with each other in common points (nodes) being the parts of two or more elements. Elements can be also connected by common boundary lines or surfaces. It can be discretized:

- The whole area of the construction (model),
- Continuous, surface and concentrated loads,
- Boundary conditions[1-5].

Only nodes are the place of interaction of elements. Continuous loads are replaced by static equivalent loads concentrated at the nodes. Thus, it can be concluded that Finite Elements Method is based on the idea of the construction of complicated objects from the simple ones or the division of large complicated objects in to smaller ones [1-4,13,16].

The analyses of the construction with the help of Finite Elements Method consists in formulation of adequate mathematical formula, and then finding solution of given problem. The condition of reliability and practical usability of FEM calculations is the ability to formulate suitable for ones needs mathematical model. It can be one dimensional, two dimensional, three dimensional and it can exist different ways of idealization of: the shape, material properties, and support and load conditions of the model.

The choice of proper mathematical model is very important step of the analysis and it determines closeness and usability to the fundamental degree. Therefore, very good knowledge of studied phenomena is the key part of efficiency of computer simulation [16-22,26].

Usually, the analytical model is constructed interactively in the one of the part of the software package called pre-processor. In this software environment, there is constructed the geometrical model of analyzed construction and material properties as well as boundary conditions are defined. After indication of the types of finite elements which will be used in the model, and requirements of division on elements, there is automatically generated the mesh of nodes and finite elements. There is possibility of automatic verifying, from the formal point of view, the discrete model of examined problem [1-5].

3. ANSYS software

ANSYS is a general purpose finite element simulation package for numerical solving a wide variety of design engineering problems. These problems include: static/dynamic structural analysis (both linear and non-linear), heat transfer and fluid problems, as well as acoustic and electromagnetic problems. There is possibility of solving mixed problems connecting the ones described above. The product includes many special tools that helps in analyzing such effects as: plasticity, large deformations, hiperplasticity, creep, high deflection, interface contact, dependence of various properties on temperature and pressure, anisotropy and radiance. For example, the possibility of substructures, submodelling and optimization are available. The ANSYS has been already used for commercial purposes since 1970. A variety of specializations under the umbrella of the mechanical engineering discipline (such as aeronautical, biomechanical, and automotive industries) commonly use ANSYS software package in design and development of their products. It is also used in electronic and power engineering, including nuclear power engineering, oil and steel industry as well as in material engineering [1-5,11-16, 23-26].

The finite element analysis package ANSYS enables engineers:

- Construction of geometrical model or reproduction of existing models already made in other commonly used CAD programmes,
- Introduction of exploitation loads or other conditions connected with the work of analyzed system,
- Analysis of stresses, temperature distribution, etc.,
- Designed system optimization at the pre-calculation stage,
- Making of such simulations of working system that are impossible to do in any other way (e.g. in biomechanical application) [1-3, 16,23-29].

In the family of ANSYS, Inc. there are:

ANSYS Workbench - complete environment dedicated for simulation and modelling.

ANSYS CFD - the modern, highly parallelized solver applied to solve wide-ranging fluid flow problems CFD, among the others CFX and FLUENT.

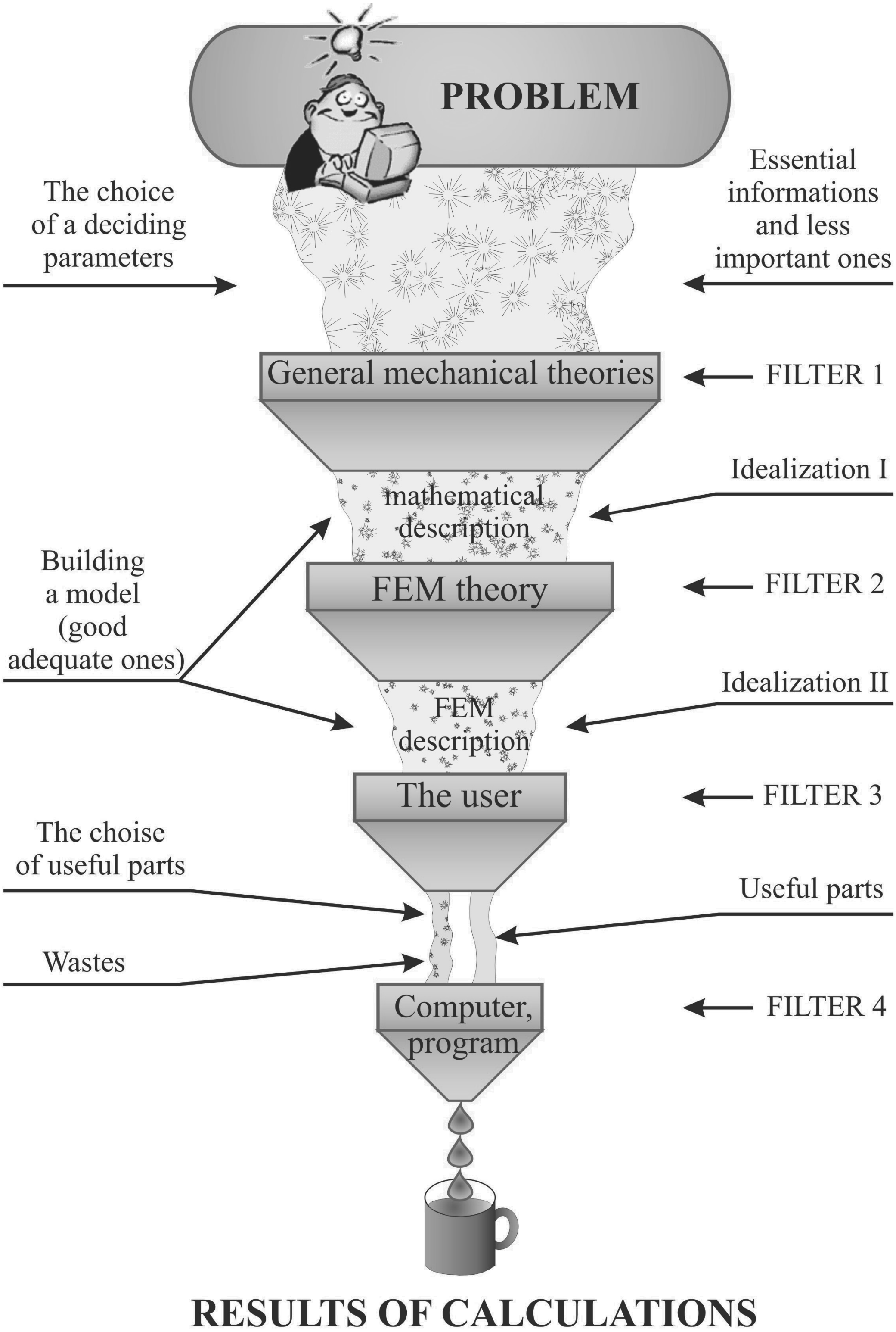

Fig. 1. Differences between real problem and results of MES calculations – scheme [22]

ANSYS AUTODYN - a uniquely versatile explicit analysis tool for modelling the nonlinear dynamics of solids, fluids, gas andtheir advanced interactions, high deformations and displacements, non-linear material behaviours, fragmentation and shock wave movements.
ANSYS LS-DYNA - LS-DYNA LASTC - an explicit dynamic solver technology with the pre-/post-processing power of ANSYS software; this powerful pairing enables analyses of nonlinear phenomena found in crash tests, metal forging, stamping and catastrophic failures.
ANSYS ICEM CFD - an advanced tool applied to discretization, with general possibilities of pre-/post-processor [2, 16-18, 20-21, 23].

ANSYS Workbench has been chosen because of its easiness to use and intuitive user interface, allowing quick and simple program operation. The ANSYS Workbench hosts many software products and components enabling for the optimal selection of options required for one's own purposes and the optimal design of the construction (e.g. with the minimal weight, suitable shape), in this way the total number of expensive prototypes is reduced as well as the time of the introduction of final product to the market. With unusual CAD connectivity, an automated project update mechanism, pervasive parameter management and integrated optimization tools, the ANSYS Workbench platform delivers productivity that truly enables product development through simulation. Some of the examples of the various and numerous ANSYS Workbench applications are listed below:

- Mechanical - applied for structural linear/nonlinear and dynamics analysis, it offers a complete set of elements behaviour, material models and equation solvers for a wide range of engineering problems including thermal analysis and coupled-physics capabilities involving acoustic, piezoelectric, thermal-structural and thermal-electric analysis.
- Mechanical APDL - dedicated for advanced mechanical and multidisciplinary analysis using traditional, classic user's interface of ANSYS.
- Fluid Flow (CFX) - a high-performance, general purpose CFD that enables capturing virtually any type of phenomena related to fluid flow.
- Fluid Flow (FLUENT) - software contains the broad physical modeling capabilities needed to model flow, turbulence, heat transfer, and reactions for any industrial applications.
- Geometry (DesignModeler) - the feature-based, parametric software that can be used to create parametric geometry from scratch or to prepare an existing CAD geometry for further analysis.
- Engineering Data - a module applied for import/export material data as XML files.
- Meshing Application - applied for CFE mesh creation and Explicite Dynamice, the product allows the user to find the balance and get the right mesh for their simulation.
- Finite Element Modeler (FE Modeler) - applied for the conversion of FEM mesh between NASTRAN ABAQUS and ANSYS programmes.
- BladeGen (Blade Geometry) - a specialized, easy-to-use tool for the rapid 3-D design of rotating machinery components [2,4,16,23].
- Explicite Dynamice - an advanced explicit analysis tool for modelling complex nonlinear dynamics.

ANSYS Workbench is the one of the most frequently used package in the world. It makes complex analysis in almost any discipline of science and industry possible. The software offers very wide spectrum o analyses ranging from statistics to electromagnetic field via phenomena variable in time and possibility of linear/nonlinear phenomena analyses. Complete analysis systems are convenient because they contain all of the necessary tasks or components to complete start-to-finish simulations starting from definition of problem, through calculation of the problem up to verification of derived results without necessity of using additional external software. Therefore, there are many prerequisites for successful applying ANSYS Workbench in material engineering as well as in many other disciplines of science and industry[23-26, 28].

4. FEM analyses application

An application of finite elements method ranges many different fields of modern science or industry. the various applications in the aircraft, space, shipbuilding, electrical and computer industry as well as field of science like material engineering, mechatronics, biochemistry and even medicine must be emphasize. Both, technological and economical pressure in product engineering require introduction of continuous optimization of existing manufacturing processes, including those in designing stage. The described method is the proper tool for these purposes. Recently, finite element method has been arousing increasing interest in medicine. Up to now, it is the widest applied in modelling of: tissues (skin, bones, and muscles), implants (stents, prostheses, elements of maxilla and teeth), and physical fields inside the body (acoustic, electric and electromagnetic ones) and the blood flow. For example, simulation of thigh bone with femoral head replaced by prosthetic implant is applied mainly for the analysis of stress, deformation and dislocation states. FEM method of simulation gives better, more accurate and wider results then experimental methods. Another example FEM application is strain analysis of joints. It allows determining the kind of stresses occurring in the bone, the location of its main occurrence. The influence of alloplastics and osteotomy for stress pattern in the bones can be also observed. The construction of the rib model is the next example of FEM simulation of bones. The analysis can be related with rib behaviour subjected to the high force of the blow, e.g. during car accident. Finite Element Method is also applied in creation of biostatic models of the mandible or one-layer aortal valve, additionally, there are realized works connected with coronary, urological or oesophageal stents [8-13, 24-29].

Considering prerequisites mentioned above, FEM appears to be the natural choice for successful simulation of internal prosthesis of oesophagus, which is described in detail in further part of this article.

5. Simulation

The shape conditions introduced for the model of internal oesophageal prosthesis followed from measurements of geometrical dimensions of biological tissues of piggy oesophagus and literature data regarding human oesophagus. At the first stage, measurements of thickness and shape of natural piggy oesophagus, that possess the dimensions closest to the human one, were conducted. On the basis of obtained results the preliminary shape of internal oesophageal prosthesis was determined. Additionally, functional features of the prosthesis as well as the technique of anastomosis between the prosthesis and the stump of oesophagus had influence on final shape of the prosthesis (Fig. 2) [7]. The geometrical model were simplified, because there were observed significant differences in shapes of investigated real piggy oesophagus.

Parameters of the model were divided into two groups:

- Shape parameters - presented in the Figure 2,
- Material parameters:
  - Young modulus (E = 100 - 120 GPa)
  - Poisson's ratio (0.4 – 0.5)

ANSYS Workbench 12.0 was used for numerical simulation. A dozen or so models with various meshing density and variable Young modulus or Poisson's ratio were done. In the presented model of internal prosthesis of oesophagus, 60568 nodes and 30577 3D SOLID types elements were generated. In the model, there were applied fixed bearings in the y and z directions, the pressure of magnitude equal to 0.2 MPa all over the length with increasing value up to the 0.5 MPa at the ends (in the place of contact between prosthesis and the stump of oesophagus) was applied and gravity force was simulated.

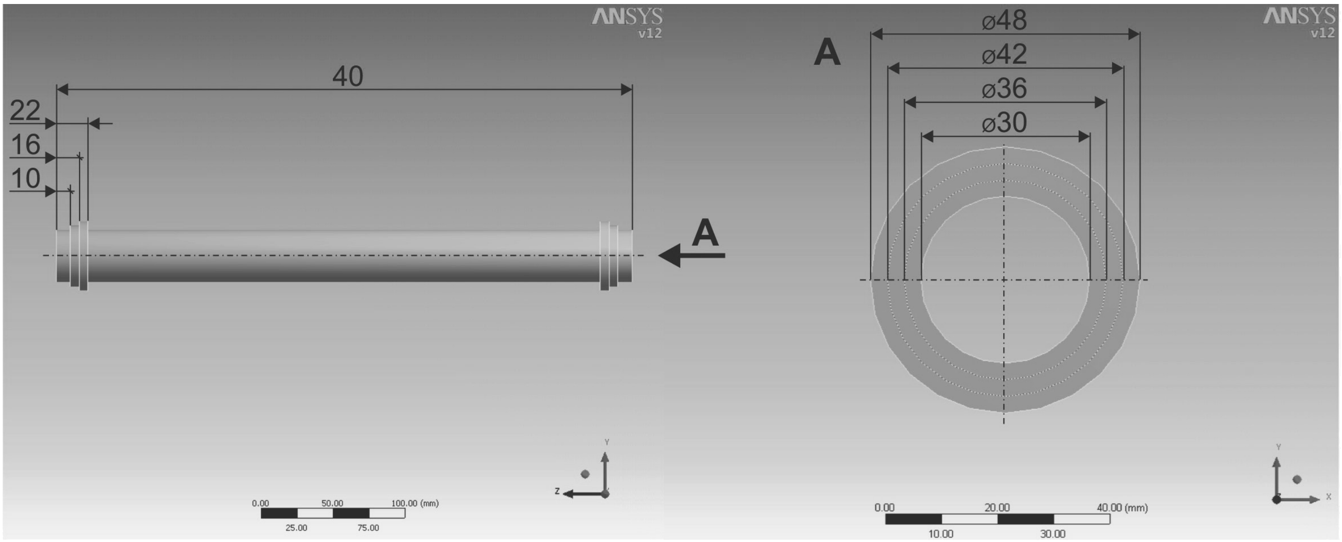

Fig. 2. Model of internal prosthesis of oesophagus

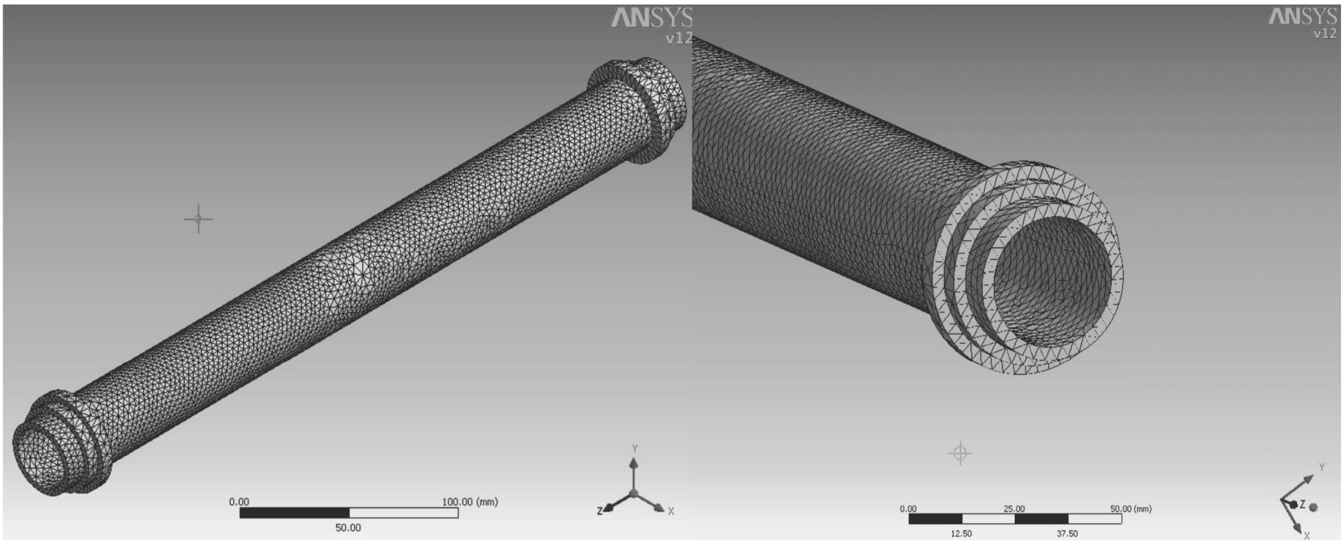

Fig. 3. The geometrical model after division into finite elements

In the Figure 4, an exemplary distribution of stresses intensity in internal prosthesis of oesophagus is presented, whereas in the Figure 5 displacements in x direction are shown. Presented results in the form of distributions of calculated values of stresses and displacements were similar for all investigated cases. The distribution of analyzed values indicates that the highest values are obtained on the internal side of the prosthesis, in the place opposite to the contact surface between the prosthesis and the stump of the oesophagus.

Analyzing obtained results for displacements distribution, it can be noticed that the highest values are also found in the place of contact between the prosthesis and the oesophageal stump that is on the rings. In theother hand, minimal displacements are observed on the internal edge of the prosthesis. Additionally, analyses of the maximal values of effective stress and x directed displacements show that the highest danger of the prosthesis inside diameter closing after implantation occurs in the place of the contact between the prosthesis and the oesophageal stump, i.e. in the place where rings which enable easy anastomosis of those elements, and in the middle part of the prosthesis.

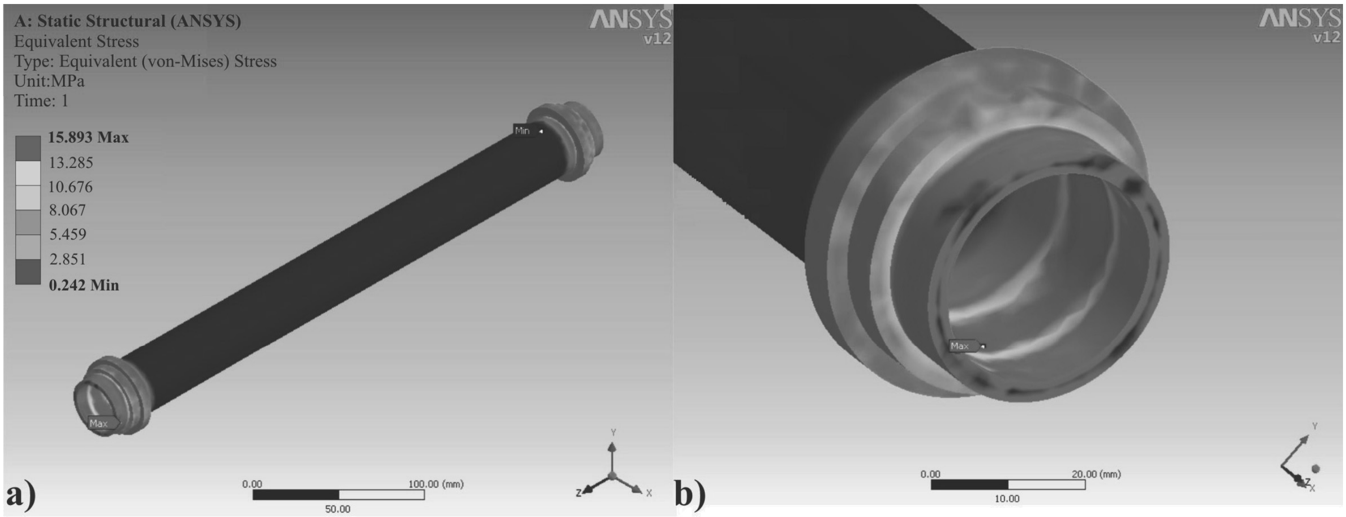

Fig. 4. Distribution of the stresses intensity a) all over the length, b) in the place of the highest stress occurrence

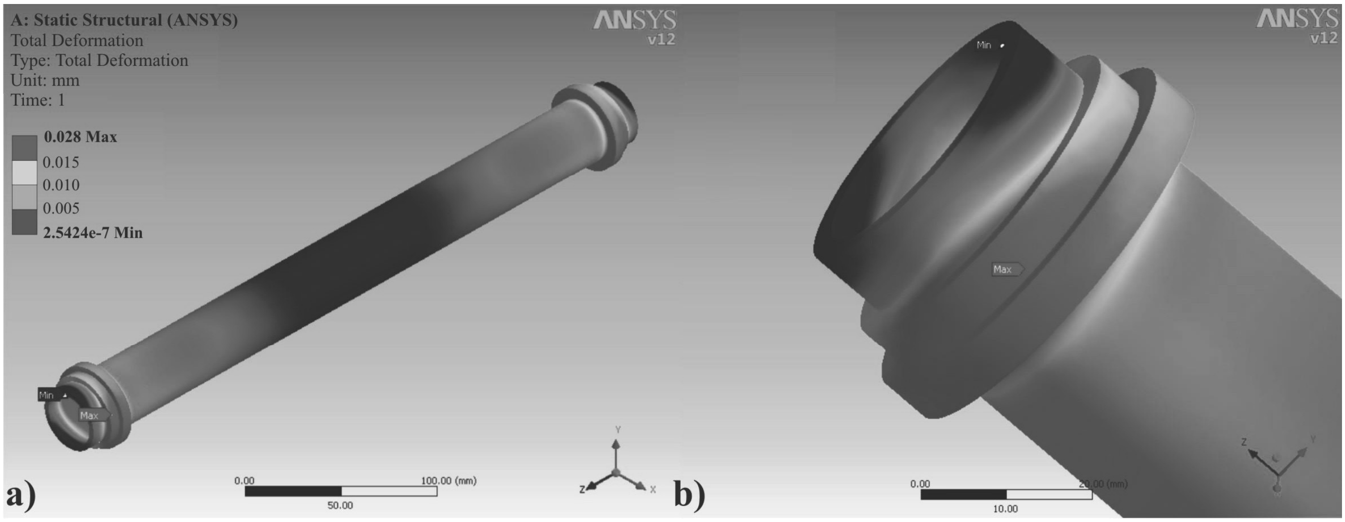

Fig. 5. Displacements values in the x direction a) for whole prosthesis, b) in the place of the maximum and the minimum occurrence

6. Conclusions

The experimental strength analysis of the internal prosthesis of oesophagus was done in order to collect data necessary for geometrical and numerical model verification, and also for assessment of correctness of applied methods adopted for simulation investigations. In the future, the analysis of three elements cooperation, prosthesis and oesophageal stumps after implantation will be done. Three variants of prosthesis implantation will be also simulated; there will be analyzed three different ways of extension of digestive system continuity including stresses of internal organs.

Carried out analysis demonstrates that the ANSYS Workbench software is a very useful tool in a various kinds of constructional elements or set of those elements design. Via discussed software, problems connected with material deformation as well as distribution of stresses occurring in it can be analyzed.

Application of computer aided materials engineering is directed on intensive progress of free and universal software, which diametrically influence on quick and reliable analysis of construction in virtual conditions, what further influence on reduction of costs and time of considered project realization.

Acknowledgements

Authors would like to thank Prof. A. Lampe and Dr M. Musiewicz from Medical University of Silesia in Katowice for cooperation.

This scientific work is financed within the framework of scientific financial resources as a research project N N507 422136 headed by Prof. L.A. Dobrzański.

Additional information

Selected issues related to this paper are planned to be presented at the 16th International Scientific Conference on Contemporary Achievements in Mechanics, Manufacturing and Materials Science CAM3S'2010 celebrating 65 years of the tradition of Materials Engineering in Silesia, Poland and the 13th International Symposium Materials IMSP'2010, Denizli, Turkey.

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 analisis, 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] T. Chmielewski, H. Nowak, Computer-aided CAD CAM, WNT - Warsaw, 1996.

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

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

[7] 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.

[8] L.A. Dobrzański, A. Pusz, 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.

[9] L.A. Dobrzański, A. Pusz, A.J. Nowak, The elimination of micropores and surface defect in aramid-silicon laminated materials with special properties, Journal of Achievements in Materials and Manufacturing Engineering, 35/2 (2010) 121-128.

[10] L.A. Dobrzański, A. Śliwa, T. Tański, Numerical simulation model for the determination of hardness for casting the magnesium alloys MCMgAl6Zn1, Archives of Materials Science 29/3 (2008) 118-124.

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

[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, Publication of Radom University of Technology, Radom, 2007 (in Polish).

[15] 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.

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

[17] P. Ladeveze, E. Le Dantec, Damage modelling of the elementary ply for laminated composites, Composites Science and Technology 43 (1992) 257-267.

[18] J. Marciniak, Biomaterials, Publication of Silesian University of Technology, Gliwice, 2002 (in Polish).

[19] J. Marciniak, Engineering of Biomaterials, Publication of J. Skalimierski Company, Gliwice, 2009 (in Polish).

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

[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] www.ansys.com
[24] T. Rojek, J.J. Telega, Numerical simulation of bone-implant systems using a more realistic model of the contact interfaces with adhesion, Journal of Theoretical and Applied Mechanics 3 (1999) 659-686.
[25] 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.
[26] 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.
[27] J. Stadnicki, Z. Tokarz, Analysis of deflection of composite aircraft wing, Bulletin of the Military Technical Academy 57/2 (2008) 45-54.
[28] Y. Takano et all., Optimizing stent expansion with new stent delivery system, Journal of the American College of Cardiology 38 (2001) 456-462.
[29] F.D. Whitcher, Symulation of in vivo loadin conditions of nitinol waskular stent structures, Computers and Structures, 64 (1997) 1007-1011.