AMME
Journal
of Achievements in Materials and Manufacturing Engineering
VOLUME 27
ISSUE 1
March
2008

Stresses present in bone surrounding dental implants in FEM model experiments

J. Żmudzki [a,*], W. Walke [b], W. Chladek [a]
[a] Department of Technological Processes Modelling and Medical Engineering,
Faculty of Materials Science and Metallurgy, Silesian University of Technology,
ul. Krasińskiego 8, 40-019 Katowice, Poland
[b] Division of Biomedical Engineering, Institute of Engineering Materials and Biomaterials,
Silesian University of Technology, ul. Konarskiego 18a, 44-100 Gliwice, Poland
* Corresponding author: E-mail address: jaroslaw.zmudzki@polsl.pl

Received 23.01.2008; published in revised form 01.03.2008

Analysis and modelling

ABSTRACT

Purpose: Appropriate selection of material and geometric characteristics of intraosseus pillar implant part creates a possibility to control the bone tissue loading.

Design/methodology/approach: A basic tool that is commonly used the evaluation of bone loading state is the linear FEM analysis. It requires setting of appropriate experiment conditions. Hence, an analysis has been carried out in order to determine the influence of dividing method of finite elements (tetragonal type 187 in Ansys system v.11) on stresses in pillar and surrounding bone tissue.

Findings: Seeking of loading values cortical bone tissue is highly affected by the increase of mesh density on the edge of implant insertion into the cortex bone. Loading stresses values have significantly increased along with increased mesh density, whereas the differences have even reach 47 MPa.

Research limitations/implications: Research has been carried out only for the Ansys system in the linear range assuming standard shape and mechanical characteristics of implant and bone, as well as regarding the after the osseointegration phase because if the presumed complete adherence of the pillar to the bone.

Practical implications: Excessive increase of mesh density leads to overestimation of loading stresses values and further to an unjustified increase of pillars' diameter. At the other hand, too large elements might lead, through an underestimation of loading stress level, to overloading atrophy of bone tissue.

Originality/value: This paper points out the necessity of more determined activities aimed on defining appropriate and uniform FEM experiment conditions that would enable achievement of more real results of model researches and their comparability.

Keywords: Numerical techniques; Dental implant; Bone; FEM

1. Introduction

One of the basic problems in biomechanical engineering is forming of implant mechanical characteristics in a way that ensures, both structure durability and an optimal load patterns in surrounding tissues [1,2,3].

A step that precedes building of prototypes for laboratory tests are researches on numerical models, which enable, at low costs, a precise definition of requirements, which the real structures should comply with [1]. A common tool used for strength model analyses is the Finite Element Method (FEM) [5,6,19,20].

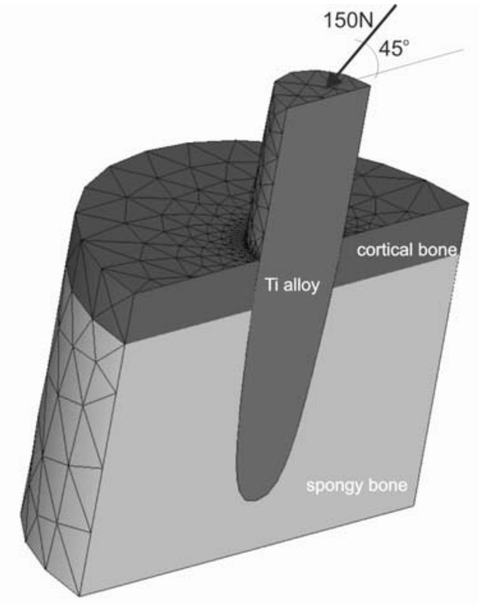

Fig. 1. Conditions of FEM model analysis

Defining appropriate model experiment conditions in models that are complex, as far as materials and geometry is concerned, especially if taking into account contact phenomena occurring in real conditions, is problematic. It is mainly due to the difficulty with gaining model convergence. Problematic is also interpretation of results because of the significant scatter of values [7]. In less complex linear models, without any contact, the problem with convergence does not exist. The existing simplicity contributes to the common use of MES in the linear aspect [8,9] in evaluation of mechanical biocompatibility in prosthetic methods of restoring organism's dexterity. It brings, however other problems connected with appropriate construction of models, which minimizes errors of that method.

In this research presented is the evaluation of loadings level in a standard implant of a denture and in the adjacent bone tissue, employing FEM (ANSYS v.11). It is aimed at defining the influence of finite element mesh parameters on scattering of stresses' values, which constitute estimation criteria in the commonly used linear analyses.

2. Methodology

ANSYS v.11 software was used in a linear-elastic range in order to carry out a FEM model analysis of bone tissue surrounding the denture. Computer parameters were as follows: CORE Quad Intel (R) Xeon(R) CPU E5335 @, 2GHz 16382GB FB DDRAM 2, FSB 1330MHz; Win Vista 64 bit. Because of the software limits only two core of the processor have been used. For the purposes of the carried out analysis bone geometry has been simplified to a cylindrical shape surrounding anchorage area. The sphere of interests is the area close to the edge at bone surface, where, as a result of stress concentration some pathological changes might take place. Hence, justified is the simplified shape of the implant without thread or any internal elements of attachment.

Model cross-section presenting the assumed system of layers is shown on Fig. 1. The pillar was loaded with a force of 150 N [10] applied at an angle of 45 degrees to implant's axis. Assumed complete adherence of the implant to the bone is reflected by the state after osseointegration phase. Model has been fixed to the bottom and side surface of the bone. Division into finite elements has been carried out at various mesh density levels close to the edge of pillar insertion into the cortex bone. Control over the size of the elements has been achieved by means of a primary division of the edge with a pre-set value of 0.1; 0.3 and 0.5. In this way models have been prepared that have various sizes of elements shown in table 1. 10-Node Tetrahedral have been used (Ansys Solid 187) [11].

Table 1.
Meshing characteristic of models

Model	Edge element length	Implant elements	Cortical bone elements	Spongy bone elements
0.1	0.1	41509	2752	3666
0.3	0.3	9024	2411	6812
0.5	0.5	2801	895	3680

Assumed were advantageous osseus foundation conditions: cortex bone thickness 2,0 mm; cortex bone's Young`s modulus of elasticity 16 000 MPa, Poisson's coefficient ν=0.3, and for a spongy bone: E=600 MPa; ν=0.4. E=110 000 MPa; ν=0,3 have been assumed as average characteristics for titanium alloys, of which pillars are to be made.

For the purposes of error estimation for the calculated stresses values, an option "energy error per element" available for linear analyses has been employed (SERR ANSYS` command) according to the method shown in paperwork [12]. By summarizing all element error energies e_i, the global energy error in the model e, can be determined. This can be normalized against the total energy (u + e), where u is the strain energy, and expressed as a percent error in energy norm, E. The percentage error in energy norm E (SEPC ANSYS` command) is indicated as a good overall global estimate of the discretization or mesh accuracy [13]. The maximum absolute value of nodal stress variation of any stress component for any node of an element (SDSG ANSYS` command) was also taken into account.

3. Results

Equivalent stresses (acc. to the Huber-Misses hypothesis) in the pillar have not exceeded 205 MPa. On Fig. 2a the pattern of equivalent stresses in cortex bone has been shown. The highest stresses level is discovered on the surface, close to the edge in the plane of a highest bending caused by the horizontal component of the loading force. Differences in stresses values between models having opposite meshes are huge. Stresses discovered in case of a low mesh density (Model 0.5) are below 62 MPa, whereas for a high density mesh (Model 0.1) they reach values above 113 MPa.

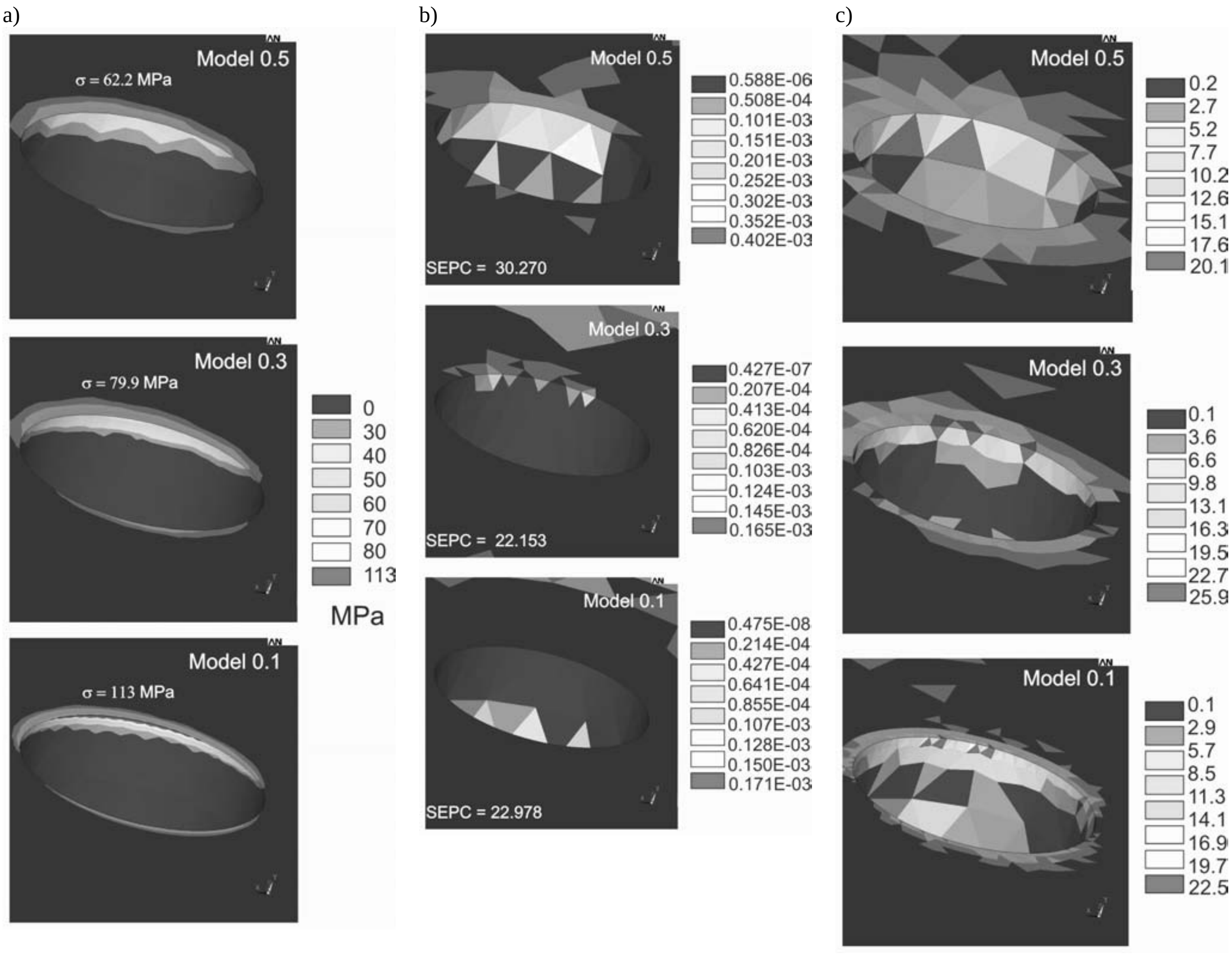

Fig. 2. The results of biomechanical analysis: a) equivalent stresses (Huber-Mises) in cortical bone for tested opposite mesh density, b) energy error per element for tested opposite mesh density, as well as an amount of structural percentage error in energy norm: SEPC, c) the maximum absolute value of nodal stress variation of any stress component for any node of an element for tested opposite mesh density

On Fig. 2b there is shown an energy error (SERR) per element and the value of structural percentage error in energy norm (SEPC) for opposite values of the tested meshes.

On Fig. 2c there is presented the maximum absolute value of nodal stress variation of any stress component for any node of an element (SDSG) for opposite values of the tested meshes.

A success of a prosthetic treatment depends directly on the risk of implant loss resulting from overloading bone tissue atrophy or fatigue implant failure, whereas pillar bending loads are in this respect very unfavorable [14, 15].

Model experiment results in an obvious way depend on a proper selection of loading and supporting conditions of models or material characteristics, which make the researchers assume some common standards in modeling, which enable not only seeking for stress levels that may occur in real systems, but also make it possible to compare the analyses.

In the modeling process, tissue geometry is usually achieved by means of spatial technique of reconstruction of 2-dimensional tomography scans. Prosthetic constructions are built directly by means of CAD systems. Usually, because of the complex shape of prosthetic structures, and at the first place bone construction, the available type of finite elements are the tetrahedral ones [8, 11].

In the analyzed model, level of stress only in the pillar of the dimension of 4mm is significantly lower than the fatigue strength of titanium alloys. It has to pointed out, however, that the results are limited to a filled single-part implant without abutments, assuming the 5mm long lateral forces lever arm.

The level of equivalent stresses for cortex tissue in the area of pillars' entrance into the bone is dangerously high. In spite of the fact that the values are located below the average cortex tissue strength, for the purposes of increasing the tolerances of cycling loads even a lot more lower values should not be exceeded in case of shear stresses of

30-35MPa [16]. Stresses result from the pillar's compression and tensions caused by bending. These phenomena take place respectively at both sides of the bent pillar. The assumed complete adherence creates situation of a symmetric stress patterns having various quantifier. In the phase before osteointegration stresses will achieve much higher level, because a portion of loadings at the side of tension cannot be borne, and the whole loading has to be borne at the compression side.

These values have been exceeded disregarding of the mesh density. Determining the real level of hazardous stresses and scopes of their occurring depends significantly on the mesh density. The problem of the influence of division results on finite elements is commonly known, however the basic rule is to carry out comparative models researches always with identical mesh parameters. Nevertheless, the seeking for real stresses values requires a detailed examination of the mesh density influence. In the presented analysis, increasing mesh density results in an increased stresses value at nodes adjacent to the edge, which constitutes in the function of stresses a special singular point because of the geometric and material notch. Differences between nodal values on the remaining length of the adherence zone are insignificant for the examined meshes.

The SEPC value, determined on the basis of SERR, significantly decreases along with increase of mesh density (Fig. 2b). Although, this parameter shows a better discretization and precision of stresses function in the whole of the bone, stresses peaks in the last of the nodes close to implant, in case of a dense mesh are characterized by a higher uncertainty (Fig. 2c). Nevertheless, it does not denote, that more precise results are achieved by a lower density of the mesh. Obtained has to be a higher precision of calculations by means of increased density, necessarily checking, at the same time not only SERR, but also SDSG parameter. One of the methods excludes those stresses values, where SDSG reaches relatively high values, and assuming as a criterion the total bone volume (or area in 2-D FEA), in which stresses exceed the assumed limit, e.g. 30 MPa. More dense mesh enables then finding a more precise isosurface separating the overloaded tissue zones from the not-overloaded. Nonetheless the extreme values should not be considered.

4. Conclusions

In software used for strength analysis, usually exposed are the extreme stresses values, although drawing conclusions on the basis of stresses peaks that occur in the area of the bone adjacent to the implants edge might lead to an invalid design. In case of a too dense mesh one gets overestimated stresses values, which leads to unnecessary increase of implant diameter. On the other hand, in case of a coarse mesh, due to the underestimated values there is a risk of causing overloading atrophy of cortex bone or formation of undesirable connected tissue [17,18].

References

[1] E. Krasicka-Cydzik, A. Kierzkowska, I. Glazowska, Effect of bending on anodized Ti6Al4V alloy: II. Behavior in vitro, Journal of Achievements in Materials and Manufacturing Engineering 17 (2006) 89-92.

[2] M. Balazic, J. Kopac, Improvements of medical implants based on modern materials and new technologies, Journal of Achievements in Materials and Manufacturing Engineering 25 (2007) 31-34.

[3] A. Ziebowicz, J. Marciniak, The use of miniplates in mandibular fractures—biomechanical analysis, Journal of Materials Processing Technology 175 (2006) 452-456.

[4] P.J. Prendergast, S.A. Maher, Issues in pre-clinical testing of implants, Journal of Materials Processing Technology 118 (2001) 337-342.

[5] F.M. Shuaeib, A.M.S. Hamouda, R.S. Radin Umar, M.M Hamdan, M.S.J. Hashm, Motorcycle helmets. Part I. Biomechanics and computational issues, Journal of Materials Processing Technology 123 (2002) 406-421.

[6] T. Mahmood, A. Mian,, M.R. Amin, G. Auner, R. Witte, H. Herfurth, G. Newaz, Finite element modeling of transmission laser microjoining process, Journal of Materials Processing Technology 186 (2007) 37-44.

[7] W.C.C. Lee, M. Zhang, X. Jia, J.T.M. Cheung, Finite element modeling of the contact interface between trans-tibial residual limb and prosthetic socket, Medical Engineering and Physics 26 (2004) 655-62.

[8] F. Nabhani, M. Wake, Computer modelling and stress analysis of the lumbar spine, Journal of Materials Processing Technology 127 (2002) 40-47.

[9] W. Wu, D.-Z. Yang, M. Qi, W.-Q. Wang, An FEA method to study flexibility of expanded coronary stents, Short technical note. Journal of Materials Processing Technology 184 (2007) 447-450

[10] S. Sahin, M.C. Cehreli, E. Yalcin, The influence of functional forces on the biomechanics of implant-supported prostheses - a review, Journal of Dentistry 30 (2002) 271-282.

[11] Ansys Element Reference, Part I. Element Library, Solid 187.

[12] O.C. Zienkiewicz, J.Z. Zhu, A simple error estimator and adaptive procedure for practical engineering analysis, International Journal for Numerical Methods in Engineering 24 (1987) 337-57.

[13] Error approximation technique for displacement-based problems. Release 10.0 Doc. for ANSYS, Chapt. 19.7.1.

[14] M. Esposito, J-M. Hirsch, U. Lekholm, P. Thomsen, Failure patterns of four osseointegrated oral implant systems, Journal of Materials Science: Materials in Medicine 8 (1997) 843-7.

[15] W. Chladek, S. Majewski, J. Zmudzki, J. Krukowska, The mechanical conditions of the functionality of choosing implant-dentures constructions – model investigations, Implantprothetic 2 (2003) 3-10 (in Polish).

[16] H. Van Oosterwyck, J. Vander Sloten, R. Puers, I. Naert, Finite Element Studies on the Role of Mechanical Loading in Bone Response Around Oral Implants, Mecanica 37 (2002) 441-51.

[17] J.B. Brunski, In vivo bone response to biomechanical loading at the bone/dental-implant interface, Advanced Dental Research 13 (1999) 99-119.

[18] S. Szmukler-Moncler, H. Salama, Y. Reingewirtz, J.H. Dubruille, Timing of loading and effect of micromotion on bone-dental implant interface: review of experimental literature, Journal of Biomedical Materials Research 43 (1998) 192-203.

[19] M. Kaczmarek, J. Tyrlik-Held, Z. Paszenda, J. Marciniak, Stents characteristics in application and material aspect, Proceedings of the 12th International Scientific Conference „Achievements in Mechanical and Materials Engineering" AMME'2003, Gliwice – Zakopane, 2003, 421-428.

[20] L. Jeziorski, J. Jasinski, M. Lubas, M. Szota, P. Lacki, B, Stodolnika, Numerical modelling of structure and mechanical properties for medical tools, Journal of Achievements in Materials and Manufacturing Engineering 24/1 (2007) 237-244.