ZESZYTY NAUKOWE POLITECHNIKI RZESZOWSKIEJ 290, Mechanika 86
RUTMech, t. XXXI, z. 86 (4/14), październik-grudzień 2014, s. 549-559

Stanisław KUT[1]

COMPARATIVE 3D FEM ANALYSIS OF THREE DIFFERENT DENTAL IMPLANT SHAPES

This paper presents the results of numerical modeling using the finite element method of three implants. Geometric models of individual parts of the analyzed system (implant, abutment and screw) and the standard models of the bone and the crown were built in the Ideas NX environment. On the basis of real geometric models the fully three-dimensional numerical models were built. The calculations for different implant systems were carried out using MARC/Mentat commercial software. The numerical models of each system consist of five deformable bodies being connected to each others. Modeling was carried out in two stages. The first stage includes the modeling of the stresses in the bone-implant-abutment-screw assembly. The preload of models was set so that the axial stress in the screw core is equal to 75% of yield stress of material from which the screw was made. In the second stage the model with assembly stresses was being loaded with oblique force on the crown with values in the range from 0 to 250 N. An analysis and comparison of stress distributions and values of stresses in analysed implant systems were carried out. This investigation shows the meaningful influence of the shape of implant of an abutment on distribution and values of stress, load capacity of individual implant systems, and furthermore, stress in osseous tissue.

Key words: dental implants, 3D numerical analysis, load, stress distribution

1. Introduction

Osseointegrated dental implants have been accepted as one of the major treatment concepts for restoring completely and partially edentulous patients over the last three decades [1-3]. These osseointegrated implants have to support the partial or full structure for dental replacement. On the other hand dental implants replace the lost physiological functions. The support of teeth and implants is inherently different. Tooth is viscoelastically supported in the bone, promoting an elastic deformation pattern, while the implant, due to its stiffness, is fairly more rigid [4]. A dental implant system consists of an implant that is surgically implanted in maxilla or mandible, and an abutment that mates with the implant

[1] Autor do korespondencji/corresponding author: Stanisław Kut, Rzeszow University of Technology, 8 Powstańców Warszawy Avenue, 35-959 Rzeszow, Poland, tel.: (17) 8651558, e-mail: stan_kut@prz.edu.pl

once the implant successfully osseointegrates to the bone. Depending on the specific system used, an abutment can include a machined connection mechanism within itself or can be clamped onto the implant by means of an abutment screw. The dental prosthesis is then fabricated over the abutment. In general, the success of the treatment depends on many factors affecting the bone–implant, implant–abutment and abutment–prosthesis interfaces [5-6]. In evaluation of the long-term treatment concept of a dental implant, there are several factors which play a great role in clinical success, such as: reliability, initial stability of the implant–abutment interface and long-term osseointegration that provides lasting incorporation into the bone and depends on implant design features such as materials, geometry and fixation methods. The last two factors are also mentioned in [4-7]. Despite the success of dental implants reported by a vast number of articles there are significant problems noticed in dental implant systems such as screw loosening and bone resorption. In order to prevent the loosening, a preload is applied to the retaining screw even that many trials are not able to eliminate its occurrence, and marginal bone resorption around implants is still unavoidable. The aim of this study was to assess the role of implant/abutment joint designs with occlusion loads transferred to the surrounding bone media using the finite element method.

2. Numerical models and materials

Three-dimensional geometric models of individual components of three different analyzed systems of dental implants performed in Ideas NX program are shown in fig. 1. Due to the plain symmetry of the analyzed systems a half of geometry was built. This allowed reducing complexities of the estimated issue and shortening the computation time, without affecting the accuracy of the results obtained by modeling. Geometric model of each system (fig. 2.) consists of bone, implant, abutment, screw, and the crown. Geometric model of the bone and the crown was made equal for all tested implant systems. Both the shape of the bone and the crown has been simplified in relation to natural one. This simplification was made because the precise mapping of the shape of the natural bone and the crown for comparative studies of different systems has no significant effect on the results, a model can greatly simplify the numerical analysis of mechanical system. While modeling the geometric models, a particular attention has paid to accurate copying of the shape and dimensions of the implant, the screw and the abutment in individual systems (fig. 1.). This is especially important for comparative studies. The tested systems chosen so as to be in the same group of measurements differ mainly in the shape of the implant-abutment connection. As a consequence of modeling, geometric models of individual components are made so that the numerical model after discretization can include as many relevant details of construction as phases, undercutting or rays. Insignificant details have been omitted during the discretization of the model.

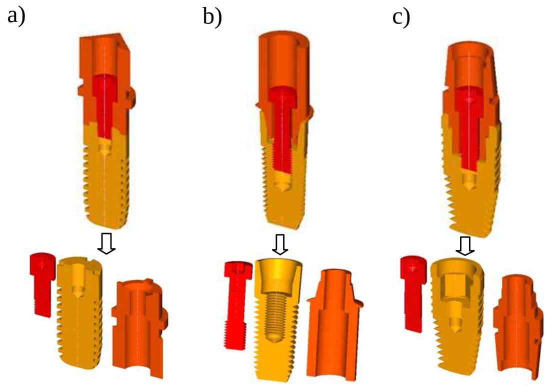

Fig. 1. Implant systems which have been chosen for the test: a) Ankylos, b) Astra Tech, c) Xive

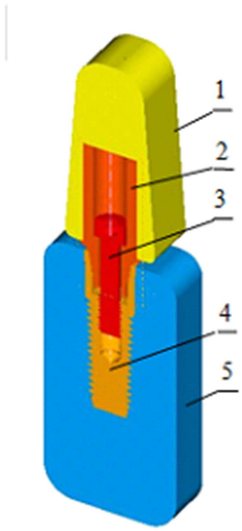

Fig. 2. The exemplary geometric model of the analysed system Astra Tech: 1 – crown, 2 – abutment, 3 – screw, 4 – implant, 5 – bone

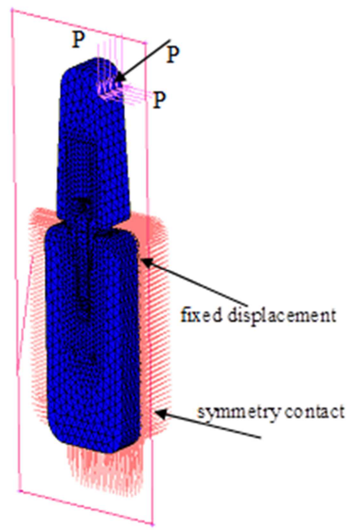

Fig. 3. Boundary conditions and the place of load application

Numerical models and calculations of studied implant systems were conducted using the MSC MARC/Mentat system. Three-dimensional, isoparametric, 10-node tetrahedron elements 127 were used to the discretization of individual parts of the numerical model [8]. Each edge forms a parabola so that four nodes define the corners of the element and a further six nodes define the position of the „midpoint" of each edge. This allows for an accurate representation of the strain field in elastic analyses. Applying 10-node elements of the second order compared with four-node elements of the first order enable more precise copy of the shape of the geometrical model and reduce the influence of the number of elements on the sensitivity of the numerical solution. This allows obtaining good results by modeling with a much smaller number of finite elements. The total number of finite elements in each numerical models was about 24 000.

Analyzed cases constitute the typical contact issue. Numerical models of individual systems consist of five deformable bodies (bone, implant, abutment, screw and crown). In all cases, glue contact type has been established between the abutment and the crown and between the implant and the bone. Between remaining surfaces of deformable bodies encountering oneself a contact of the type touching was accepted described with Coulomb's law. The value of the friction coefficient between these surfaces was assumed $\mu = 0.36$ [9]. Boundary conditions have been assumed so that from the outermost the transfer of extreme nodes of bone was blocked, while in the plane of symmetry a symmetric contact was assumed (fig. 3.). In all cases, the system was loaded by force P applied to the crown at an angle of 45^o in the range from 0 to 250 N. Place and manner of application of the load to the model in the form of the force P values $P_x = P_y = P \cos 45^o$ are presented in fig. 3.

Knowledge of materials, from which they were made individual elements of implant systems and their mechanical characteristics, is the basis of modeling success. In practice, dental implants are manufactured from titanium alloy. Commercial pure titanium, Ti-6Al-4V and Ti-6Al-4V ELI have basically been developed for structural materials although they are still widely used as representative titanium alloys for implant materials. Recently, V free $\alpha + \beta$ type alloys such as Ti-6Al-7Nb and Ti-5Al-2.5Fe have appeared as implant materials [10]. In addition, V and Al free $\alpha + \beta$ type alloys composed of non-toxic elements like Ti-15Sn-4Nb-2Ta-0.2Pd and Ti-15Zr-4Nb-4Ta-0.2Pd have been developed [11]. Low modulus alloys are nowadays desired because the module of alloys is required to be much more similar to that of bone. The β typ alloys have been, therefore, developed or are developing mainly in the USA [12]. They are composed of non-toxic elements like Nb, Ta, Zr and so on. Pure titanium and Ti-6Al-4V type alloys are also the main implant materials in the dental field. However, titanium alloys used as dental implant materials are the same as those for surgical implant materials. The alloys for other dental usage like crown, clasp and so on has somewhat different compositions compared with those for surgical implant materials except for Ti-6Al-4V and Ti-6Al-7Nb. They are in general processed by casting and superplastic forming.

The aim of these conducted investigations is comparing the influence of the shape of different systems of dental implants. For this reason they assumed that individual elements of all studied systems were carried out for the same materials. They assumed that the implant, the abutment and the screw had been made of titanium Ti-6Al-4 V alloy [10]. For these elements to modeling, an elasto-plastic material model without strain hardening phenomenon was adopted. For remaining elements (the cortical bone, the cancellous bone and the crown) an elastic model of material was accepted. Mechanical properties of individual materials used in modeling are presented in the tab. 1.

Table 1. Material properties used in the finite element model

Element of the model	Young's modulus [MPa]	Poisson's ratio	Yield stress [MPa]
Implant, abutment, abutment screw [10]	110000	0.3	729
Crown [13]	66900	0.29	-
Compact bone [14]	13760	0.3	-
Cancellous bone [14]	7930	0.3	-

Preliminary stresses are playing a very important role in correct functioning of the implant system, incurred as a result of tightening the implant to the abutment clamped by means of an abutment screw. Too high value of the initial stress can affect the bone tissue which is very unfavorable. One of the main complications is the inflammation of the peri-implant tissues called peri-implantitis. Moreover the high value of preliminary stresses in combination with stresses originating from occlusal forces can trigger inadmissible permanent plastic deformations of the abutment screw or the implant. On the other hand, excessively low values of assembly stress can lead to the relief of combined elements under the influence of variable loads, and hence to their destruction. Therefore it is very important at the time of mounting the torque. So that to make the implant-abutment joint correct, the screw should always be initially stretched irrespective of occlusal forces (chewing forces). While burdening the implant system, the preliminary tension in the screw should not fall to zero or did not cross the border of the screw's plasticity. The load of the screw depends on the type of the abutment. Conducted investigations [13] showed that optimal preliminary loading of the screw after the assembly should be 75% yield stress of material from which it is made. Therefore, the strength of screws preload of each system established so that the initial stress in the core was about 547 MPa. For example, the screws with a diameter of core of 1.2 mm (Astra Tech system) the value of preload force was 618 N.

While modeling preliminary stresses in the system were being triggered to this purpose exploiting abilities of the contact of the type glue. For this purpose the screw was divided into two deformable bodies, and a crack was created between them. After starting calculations in the first step surfaces are moving to themselves and then they are being glued together, what in the analyzed arrangement assembly stresses are arising as a result of it. The size of gap was such that the surface of the tightening of screw surface induces stress in the core of the required values indicated above.

3. FEM simulation results

Performed numerical modeling allowed to appoint disintegrations and values of stresses appearing in individual areas of component parts of studied implant systems. A comparative analysis of stresses was carried out for individual parts of the system. First an analysis of the effect of the type of the implant system on bone tissue was conducted. From the relation between the effective tension and the load *P* (fig. 4.) results, that the impact of the type of the implant system on the size of stresses which are transferred to the bone is very significant. The figure 5. presents the values of stresses transferred to the bone in the most loaded site A. For the load *P* = 0 N initial stress occurring in the system is a result of an implant abutment joint tightened with a screw. Preload (assembly) all the time impacts on bone regardless of the load of external occlusal forces. Therefore, their value is very important and is approximately 13.5 MPa for Xive, 10.5 MPa for Ankylos and 8.5 MPa for Astra Tech. After applying load *P* the value of stresses increases. The largest increase in the stress value was observed for the Astra Tech (fig. 4. – red line), and lowest for Xive (green line). In all cases, the tension rising in proportion with the load. When the load *P* = 100 N the intensity of stress in the most loaded place of bone is for Ankylos 69%, and for Astra Tech system up to 153%, higher in comparison with Xive system.

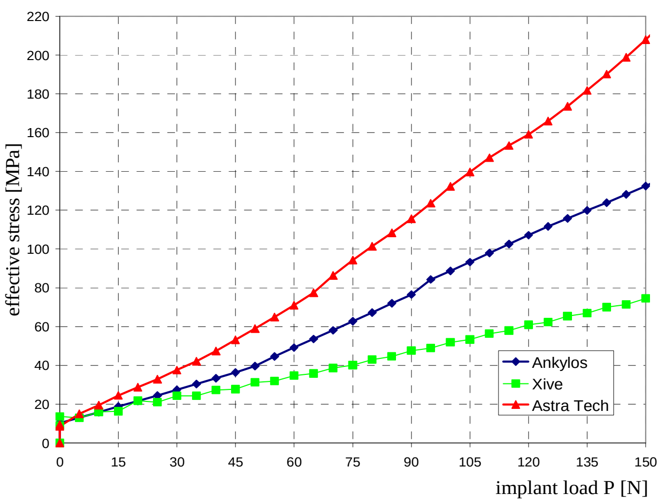

Fig. 4. The relation between the effective stress transferred to the bone and the load for studied systems

Further comparative analyses of disintegrations and stresses values in abutments of implants and individual screws of implant systems were carried out using the non-dimensional standardized tension in the form of the indicator: k = (effective stress/yield stress).

If ($0 < k < 1$) the material deforms in the elastic range. If $k = 1$ then the substitutive load reaches a value of plasticizer load equal to yield of material of which the structural element is loaded. In cases where $k > 1$ the material plastically deforms, which leads to permanent deformation of structural elements. In endurance analyses of structural elements the k indicator with the help of one numerical value is announcing the possibility of using the carrying capacity of individual elements of the structure. In order not to allow to the material plasticizing, and hence causing long-lasting deformations of the structure oneself, the value of an indicator should be $0 < k < 1$. Therefore, the use of such a normalized stress index k is very comfortable, which was used in the analysis of research results presented in this work.

Figure 5. presents the stress distribution and the values of normalized index k in the abutments of analyzed systems after a load of P = 200 N. One can see the large impact of component force P_x causing bending of the system and consequently stress concentration on the values of $k \approx 1$ (Ankylos and Astra Tech systems). In case of the Ankylos system relatively the small size of the implant abutment interface caused the great concentration of stresses which is biggest in the action area of stresses from the bending force (yellow color). This is the most dangerous place in the abutment under load so that it can be plasticized. It is similar to the case of an implant abutment joint of Astra Tech system. Stress from bending forces is concentrated, and this pattern of force can lead to permanent deformation of the abutment in the area of greatest stress values (yellow color). Much better situation is in implant abutment joint of the system Xive. In this case, the indicator k does not exceed the value of 0.7 (fig. 5.).

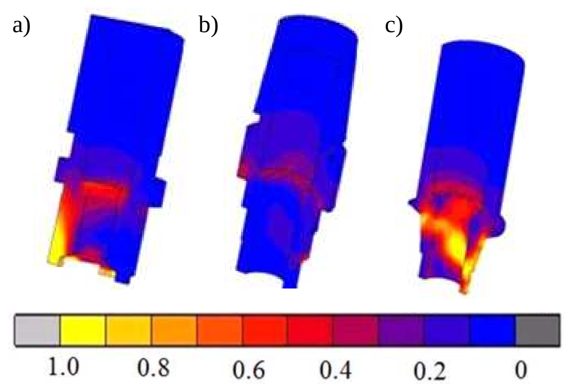

Fig. 5. Comparison of distributions of the normalized stress index k in abutments after load (P = 200 N) for different implant systems: a) Ankylos, b) Xive, c) Astra Tech

Implant abutment connection form is designed so that both the load from the component axial force P_y and the bending force distributed on a large surface area of contact which substantially reduces the level of stress. Similarly is in the case of implants, in which the stresses are a result of the impact of load applied to the crown and the connecting screws (fig. 6.). As well as, this case may be areas plasticized with the greatest values of stress, which are loaded with force P = 200 N have already reached the value of locally $k \approx 1$ in the Ankylos and Astra Tech systems. In addition to the shape of the implant connection with the abutment, a screw has the significant impact on capacity of the system, whose task is to merge with the rest of the implant system, triggering and keeping preliminary stresses (assembly). Initial stresses in the screw cores should be large enough so that while loading tooth crowns with the strength of the axial stress is not reduced to zero. The absence of this condition may lead to loosening the connection, which is unacceptable.

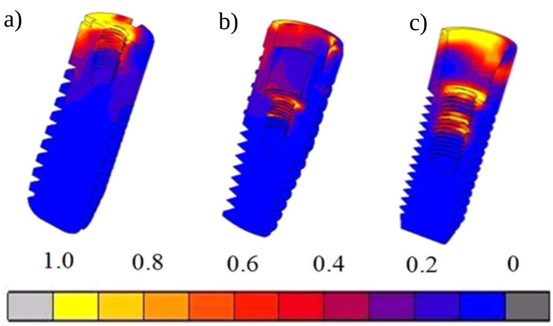

Fig. 6. Comparison of disintegrations of the normalized stress index k in implants of individual systems: a) Ankylos, b) Xive, c) Astra Tech; load P = 200 N

Figure 7. presents the stress distributions and deformations of examined implant systems under the impact of load values P = 200 N. Ankylos system is the least resistant to the transfer of bending forces. The weakest element in this system is a screw, which is the earliest part that plasticized, with power $P \approx 100$ N. Astra Tech system (fig. 7c), compared with Ankylos system (fig. 7a), demonstrates greater resistance to lateral forces. In this case, stress from the bending is transmitted by the conical surfaces. However, increasing the power $P \approx 200$ N may plasticize most loaded areas (yellow), and consequently lead to the deformation of the system. The most resistant to the transfer of both axial and lateral loads is Xive system (fig. 7b). In this case, the load P = 200 N does not cause significant deformation of the system, and stress values are below the yield point ($k < 1$). The shape of the abutment and the implant cause that forces

transmitted by the screw is considerably smaller than in the other analyzed systems.

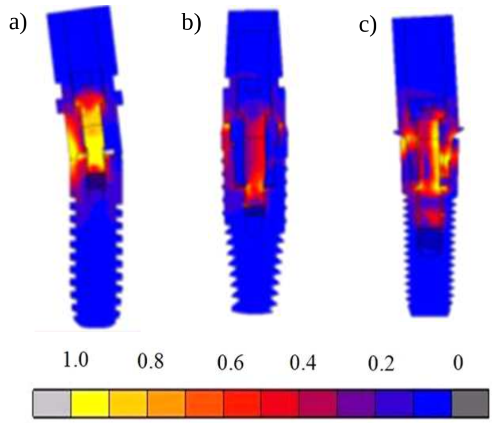

Fig. 7. Comparison of the deformation and distribution of the normalized *k* index for different values of loads for individual systems: a) Ankylos, b) Xive, c) Astra Tech

4. Conclusions

These studies demonstrated that in case of oblique load most beneficial both in the aspect of applied load on bone tissue, as well as the carrying capacity and the stiffness is implantological system Xive. It can transfer loads a little bit above 200 N without significant deformations. The shape of the connector in this system is so designed that big gradients of stresses do not appear. However it has the most complex shape of the connector, which can influence on inaccuracies of performing connection between the implant and the abutment. Ankylos system can transmit high axial loads, but is the least resistant to lateral loads. Applied oblique power with the $P \approx 100$ N value can trigger permanent deformations in this system. The weakest element in this system is a screw, which first can be subjected to yield and trigger further permanent deformations of the system. The Astra Tech transfers both axial and lateral forces. However, the conical shape of the connector can, at great axial loads cause stretching the cone and thus it reduces the preliminary tension triggered by the screw. That in consequence may lead to implant connection loosening. Lateral forces cause considerable stresses concentration and straining the screw, moreover it significantly reduces the carrying capacity of the system, which for the modeled coincidence was slightly less than 150 N.

References

[1] Chun L.L., Kuo Y.-C., Lin T.-S.: Effects of dental implant length and bone quality on biomechanical responses in bone aound implants: A 3D non linear finite element analysis. Biomed. Eng. Appl. Basis Comm., 17 (2005), 44-49.
[2] Sahin S., Cehreli M.C., Yalcin E.: The influence of functional forces on the biomechanics of implantsupported proethses – a review. J. Dent., 30 (2002), 271-282.
[3] Tada S., Stegaroiu R., Kitamura E., Miyakawa O., Kusakari H.: Influence of implant design and bone quality on stress/strain distribution in bone around implants: a 3-dimensional finite element analysis. Int. J. Oral Maxillofac. Implants, 18 (2003), 357-368.
[4] Carvalho L., Ramos A., Simões J.A.: Finite element analysis of a dental implant system with an elatomeric stress barrier. Summer Bioengineering Conf., Florida 2003.
[5] Bozkaya D., Muftu S.: Mechanics of the tapered interference fit in dental implants. J. Biomech., 36 (2003), 1649-1658.
[6] Geng J., Tan K.B.C., Liu G.: Application of finite element analysis in implant dentistry: a review of the literature. J. Prosthetic Dentistry, 85 (2001), 585-598.
[7] Waters N.E.: Some mechanical and physical properties of teeth. Symp. Society for Experimental Biology, Cambridge University Press, London 1980, vol. 34, 99-135.
[8] MSC Software: MSC. Marc Volume B: Element Library, Version 2010.
[9] Machinerys Handbook Eighteenth edition, Kempes Engineers Year Book 1980.
[10] Niinomi M.: Mechanical properties of biomedical titanum alloys, Mat. Sci. Eng., A243 (1998), 231-236.
[11] Iron and Steel Institute of Japan. Data Book of Fracture Toughness of Titanium Alloys, Tokyo 1990.
[12] Wang K.: The use of titanum for medical applications in the USA. Mat. Sci. Eng., A213 (1996), 134-137.
[13] Lang L.A., Kang B., Wang R.F., Lang B.: Finite element analysis to determine implant preload. J. Prosthetic Dentistry, 90 (2003), 539-546.

BADANIA PORÓWNAWCZE TRZECH RÓŻNYCH KSZTAŁTÓW IMPLANTÓW DENTYSTYCZNYCH Z ZASTOSOWANIEM 3D MES

Streszczenie

W pracy przedstawiono wyniki modelowania numerycznego MES trzech systemów implantologicznych. Modele geometryczne poszczególnych części badanych układów (implant, łącznik, śruba) oraz standardowe modele kości i korony zostały utworzone w programie Ideas NX. Na podstawie modeli geometrycznych zbudowano trójwymiarowe modele numeryczne badanych systemów. Symulacje numeryczne poszczególnych systemów implantologicznych przeprowadzono za pomocą komercyjnego oprogramowania MARC/Mentat. Model numeryczny każdego systemu składał się z pięciu ciał odkształcalnych połączonych ze sobą. Analizę numeryczną dla poszczególnych systemów implantów przeprowadzono w dwóch etapach. Pierwszy obejmował modelowanie naprężeń montażowych. Obciążenie montażowe było zadawane w taki sposób, aby naprężenia osiowe w rdzeniu śruby wynosiły 75% granicy plastyczności materiału, z którego jest ona wykonana. W drugim etapie model ze wstępnymi naprężeniami montażowymi był obciążany na koronie ukośnie siłą w zakresie wartości od 0 do 250 N. Dokonano analizy oraz porównania rozkładów i wartości naprężeń występujących w badanych systemach implantologicznych. Badania wykazały znaczący wpływ kształtu implantu oraz łącznika na rozkład i poziom naprężeń oraz nośność poszczególnych systemów implantologicznych, a także na naprężenia w tkance kostnej.

Słowa kluczowe: implanty dentystyczne, analiza numeryczna 3D, obciążenie, rozkład naprężeń

DOI: 10.7862/rm.2014.58

Otrzymano/received: 20.07.2014 r.
Zaakceptowano/accepted: 22.11.2014 r.