MODELARE NUMERICĂ 3D CU METODA ELEMENTELOR FINITE PENTRU BIOMATERIALE FOLOSITE ÎN IMPLANTELE DENTARE

THREE DIMENSIONAL FINITE ELEMENT METHOD MODELING OF DENTAL IMPLANTS BIOMATERIALS

IULIA ROATEŞI*

[1]*University of Medicine and Pharmacy Carol Davila, 8 Eroii Sanitari Bd., 050474, Bucharest, Romania*

Dental implants (DI) are biocompatible structures made of alloplastic materials that are inserted surgically in the bone crest to become outstanding infrastructure for prostheses or dentures. Materials used in DI manufacturing are represented mainly by titanium and its alloys, but can be also manufactured by zirconia, safirbioceramics, etc. This article is dedicated to a 3D finite element modeling of various types of DI to calculate the stress and distribution of safety factor under masticatory forces, and therefore to assess risk factors for DI design. This work represents an original study of accurate geometric models of various types of DI and therefore for their using in calculation to evaluate the risk zones in the whole structure made up of bone, implant and ceramic crown.

Implantele dentare (ID) reprezintă structuri biocompatibile realizate din materiale aloplastice care se inseră chirurgical la nivelul crestelor osoase restante pentru a deveni infrastructuri pentru lucrări protetice. Materialele din care se confecţionează ID sunt reprezentate în principal de titan şi aliajele acestuia, dar pot fi fabricate şi din zirconiu, safirbioceramică, etc. Acest articol este dedicat modelării 3D prin metoda elementelor finite (MEF) a diverse tipuri de ID pentru a calcula starea de tensiune şi distribuţia factorului de siguranţă sub acţiunea forţelor masticatorii, şi în consecinţă a evalua factorii de risc în cazul proiectării acestora. Acest articol reprezintă un studiu original prin realizarea de modele geometrice de mare acurateţe pentru diverse tipuri de ID şi folosirea lor în calcule privind evaluarea zonelor de risc în structura formată din os, implant, coroană ceramică.

Keywords: *dental implants, biomaterials, bioceramics, three-dimensional finite element analysis, numerical modeling, implant stability*

1. Introduction

Biomaterials research is an important topic at present, together with the rapid development of dental implantology.

A prerequisite imposed on all biomaterials is to provide local and general innocuous. Avoid materials that have toxic, carcinogenic, allergic and/or radioactive. In general, biomaterials must be biologically, mechanical, functional compatible, corrosion resistant, and easy to adapt clinical and laboratory technologies [1].

In terms of scientific and practical point of view, first occupied as materials for DI are metal alloys as they posses high resistive properties (compressive strength, flexural strength, etc.) in order to retrieve and transmit the forces exerted at this level to the physiological bone.

While the 70s were used Co-Cr-Mo alloys and tantalum as materials for DI, currently preferred implants made of pure titanium and titanium alloys.

The chemical and biological properties of titanium are dictated by the oxide surface layer. One of the properties that distinguish titanium from the other metals that are within the scope of biomaterials is the physicochemical stability during casting.

DI made from aluminous ceramic (Frial, Biolok, Bion) were the first achievements in the field. Ceramic aluminum oxide differs essentially from metal. Thus, the aluminous ceramic implants have a very high hardness that allows any processing only with diamond tools, running water and a compressive strength far above the metal implants.

DI made from ceramic based on zirconium oxide can be included in implant category of endosseous stabilization of periodontal teeth. They have adequate mechanical strength and recognized biocompatibility. The rods are inserted proximally with respect to the natural teeth.

Light ceramics and those based on zirconium oxide produce contact osteogenesis, so around the implant will be submitted lamellar bone, mechanically durable.

Lately try marketing of DI made solely of zirconia. It is possible that in the future these DI even replace those made of titanium, but clinical trials are currently insufficient.

DI may be several forms, some of them historical, others successfully used today. The most common types of implants and which are mainly studied in this article are the implant type of the

---

* Autor corespondent/*Corresponding author,*
E-mail: iulia.roatesi@gmail.com

root (Root form implants) that can be cylindrical or screw. Less used implants are blade type, blade-cylinder type implants, transmandibulare or transfixante implants.

This article presents a study of DI using numerical models by FEM. This method [2-4] is widely used in the study of dental implants and proved very useful in analyzing the distribution of stress and strain in the whole structure made of bone, implant and crown [5-9]

These studies are relevant in achieving optimal design of the implant [7-9], in the study of osseointegration [10], in the study of oral rehabilitation [11,12], by different options of implant loading and orientations, etc.

Numerical analysis of the implants is a simulation of clinical situations and it is useful for a thorough and detailed understanding of the characteristics of stress and strain in the implant and in the surrounding bone structure to ensure the success of implants.

The importance of numerical analysis by FEM in the study of dental implants involves several aspects.

It is equally useful both to clinicians, by investigating alternative treatments and to DI manufacturers. They change their macro-design and bonds in agreement with the clinical benefits seen. To improve the design and use of DI is the absorption as reduced as possible of the bone in the region around the implant, a reduced micro-movement of the abutment, a better load distribution in implant structures, a good cone sealing. All these properties are often related to the biomechanical behavior and they should be investigated not only in clinical trials but also in studies of FEM.

A new design of DI and materials should be subjected to a thorough investigation and to be compared with traditional structures. FEM modeling analyzes allow the comparison of old and new treatment, taking into account the limitations and deeper understanding of areas of application.

2. Material and method

The present study is dedicated to the presentation of some precise geometrical models for different types of DI, with a high accuracy of all tecnical details of the structure.

Some of these models are used to calculate by Cosmos program in order to simulate numerically some complex clinical situations, namely the insertion of an implant in a section of the mandible.

On this structure, made up of mandible, implant, abutment and eventually ceramic crown, loads caused by the mastication forces are applied. These loadings determine stress concentration areas. These areas are the most vulnerable to occur possible material damage, failure or rupture.

Since achieving the geometric model of the dental implant, crown and surrounding bone requires special preprocessing resources, Solid Works program was used [13]. The geometric model done with this program was exported and used by Cosmos program [14] for the calculations of the implant-bone structure.

2.1. The geometric models for different types of dental implants

Figures 1-11 present the geometric models for different types of implant. These models were made on the computer with SolidWorks software, precisellly respecting all sizes and technical details of a real implant.

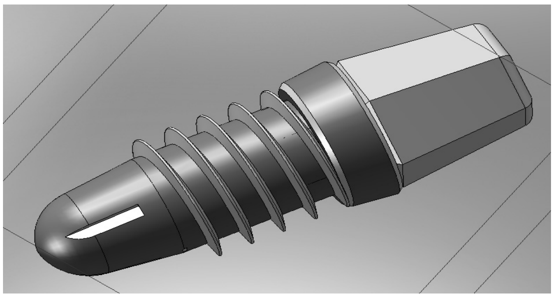


Fig. 1 - Geometric model of the implant type1 / *Model geometric al implantului tip 1.*

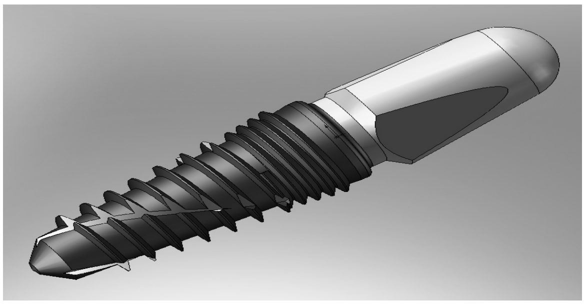


Fig. 2 - Geometric model of the implant type 2 / *Model geometric al implantului tip 2 .*

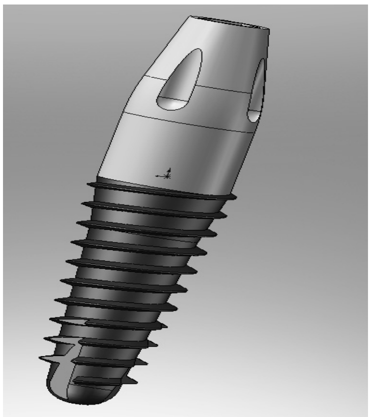


Fig. 3 - Geometric model of the implant type 3 / *Model geometric al implantului tip 3 .*

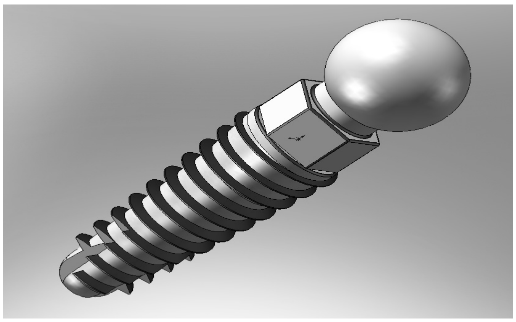


Fig. 4 - Geometric model of the implant type 4 / *Model geometric al implantului tip 4 .*

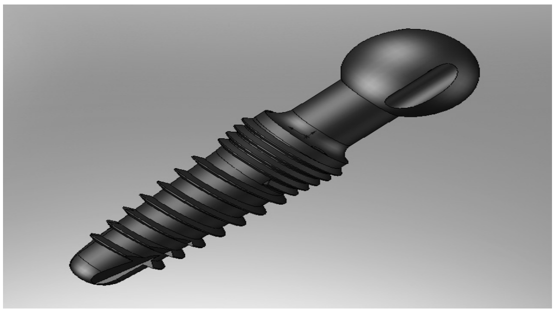


Fig. 5 - Geometric model of the implant type 5 / *Model geometric al implantului tip 5.*

To view the modeling of the inner part of implants, we present the following figures. Thus, figures 6-9 represent the implant overview, the implant sectional overview, the abutment and the abutment sectional view, respectively.

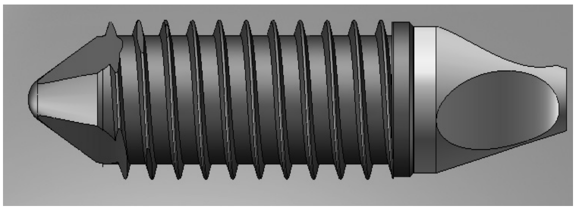


Fig. 6 - Geometric model of the implant type 6 / *Model geometric al implantului tip 6.*

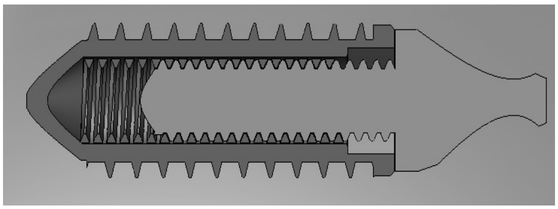


Fig. 7 - Section of geometric model of implant type 6 / *Secțiune model geometric al implantului tip 6.*

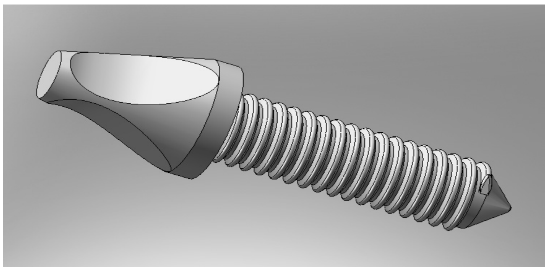


Fig.8 - Geometric model of type 6 implant abutment / *Model geometric al bontului implantului tip 6.*

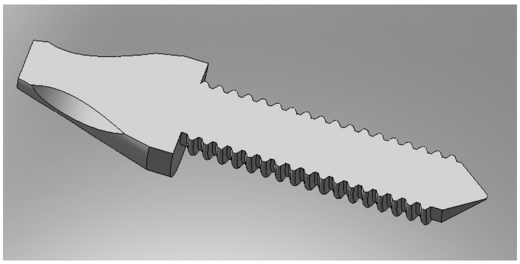


Fig.9 - Section of geometric model of type 6 implant abutment / *Secțiune model geometric al bontului implantului tip 6.*

The following types of implants are commonly used due to their shape and design resulting in a better transmission of masticatory forces with a good adaptation. Figures 10, 11 and 12 respectively represent geometric model of a Needle Denti implant; the geometric model of Rootform implant and the crown; and finite element model (EF) model of the Rootform implant and abutment.

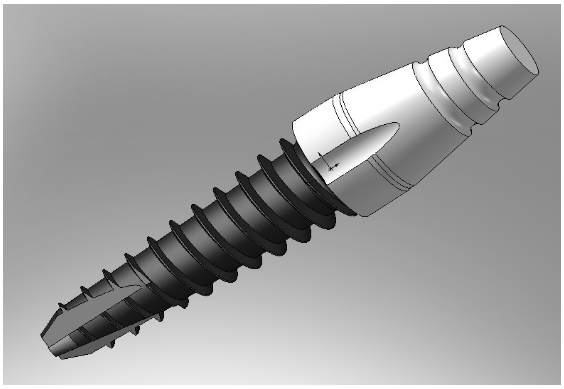


Fig.10 - Geometric model of implant Denti Needle / *Model geometric al implantului Denti Needle.*

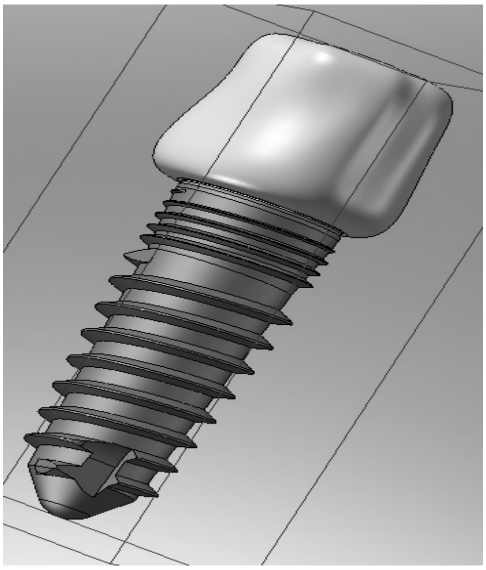


Fig.11 - Geometric model of implant Rootform and the crown / *Model geometric al implantului Rootform cu coroana ceramică.*

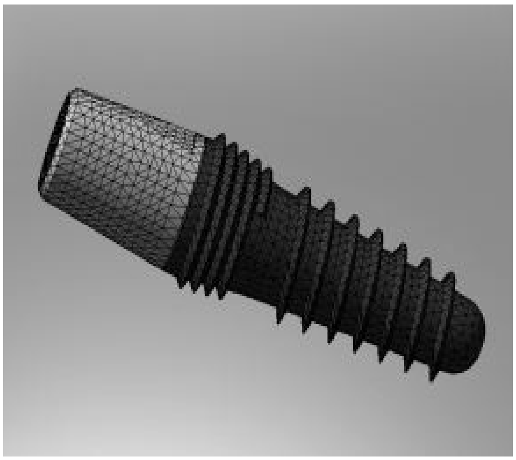


Fig.12 - Geometric model of implant Rootform / *Model geometric al implantului Rootform.*

2.2. The FE model of dental implant

The 3D model used for the study of an implant in a mandible portion was built using SolidWorks software and uses tetrahedral elements both in implant and bone, see Figure 12.

In Table 1 we present the characteristics of the numerical study.

As it is seen in table 1, the mesh contains a large number of finite elements due to the fact that the DI posses an extremely fine structures like threads, ridges, etc.

2.3. The types of material and material constants

Numerical calculations were performed for different types of materials coresponding to each structure component, i.e. bone (consisting of trabecular bone and cortical bone), implant and crown.

The data concerning the materials are presented in the tables as follows: in Table 2, the type of material for each structure component, the mass and volume; Table 3 indicates the material constants used, respectively, to the implant, its components and the ceramic crown; and Table 4 indicates the material constants for trabecular and cortical bone. This data are taken from the material library of Solid Works program [13] and they were used to create some other numerical models in [6].

2.4. The contact modeling

The threaded portion of the implant, mandible and screw represent contact areas and they are modeled with finite elements, special contact elements, determining a model according to a real behavior.

2.5. Boundary conditions and loading application

Boundary condition are set in terms of displacements and / or structural forces in those regions that are considered to be fixed during the simulation, or having specified values for displacements and / or forces.

Restrictions of null diplacement are put on some borders of the model to obtain the balance of the solution. Also, the restrictions should be placed in nodes that are away from the region of interest, in this case, the area surrounding the implant. It does so, in order to prevent overlapping of stress or strain field associated with the reaction forces of the bone-implant interface.

Table1

Mesh Information/ *Informații asupra rețelei*

Element size/*Mărimea elementului*	0.4 mm
Tolerance/*Toleranța*	0.02 mm
Number of elements/*Număr de elemente*	186216
Number of nodes/*Număr de noduri*	217356
Time to complete mesh/*Timp de realizare a rețelei(hh;mm;ss)/*	00:02:03

Table 2

Material type and their characteristics / *Tipurile de material şi caracteristicile lor*

Nr.	Component name *Numele componentei*	Material	Mass / *Masa*	Volum
1	Crown/ *Coroana*	Ceramic /*Ceramică*	0.000223658 kg	9.72426e-008 m^3
2	Abutment/ *Bont Protetic*	Magnesium Alloy *Aliaj de magneziu*	8.17675e-005 kg	4.08837e-008 m^3
3	Trabecular bone *Os Trabecular*	Trabecular bone *Os Trabecular*	0.00162148 kg	1.08099e-006 m^3
4	Cortical bone/*Os cortical*	Cortical bone *Os cortical*	0.000677504 kg	3.38752e-007 m^3
5	Implant 3.8x11.5	Titan Alloy Ti6 Al-4VS *Aliaj Titan*	0.000436557 kg	9.85725e-008 m^3
6	Screw/*Şurub*	Titan Alloy Ti6 Al-4VS *Aliaj Titan*	0.000172112 kg	3.88621e-008 m^3

Table 3

Material constants of the implant, its components and crown
Constantele de material folosite pentru implant, componentele sale şi coroana ceramică

	Magnesium Alloy (Intermediate part) *Aliaj de magneziu* *(Piesa intermediară)*		Titan Alloy (Implant and screw) *Aliaj de titan* *(Implant şi şurub)*		Ceramic (Crown) *Ceramică* *(Coroana)*	
Constant name ***Numele constantei***	**Value** ***Valoare***	**Unit** ***Unitate***	**Value** ***Valoare***	**Unit** ***Unitate***	**Value** ***Valoare***	**Unit** ***Unitate***
Elastic modulus *Modulul elastic*	$4.2\cdot10^{10}$	N/m^2	$1.048\cdot10^{11}$	N/m^2	$2.2059\cdot10^{11}$	N/m^2
Poisson Coefficient *Coeficientul Poisson*	0.33		0.31		0.22	
Shear modulus *Modul de forfecare*	$7.\ \cdot10^{10}$	N/m^2	$4.1024\cdot10^{10}$	N/m^2	$9.0407\cdot10^{10}$	N/m^2
Mass density *Densitate*	2000	kg/m^3	4428.8	kg/m^3	2300	kg/m^3
Tensile strength *Rezistența la tracțiune*	$4.\ \cdot10^{8}$	N/m^2	$8.2737\cdot10^{8}$	N/m^2	$1.7234\cdot10^{8}$	N/m^2
Yield strength *Limita de plasticitate*	$1\cdot10^{8}$	N/m^2	$1.05\cdot10^{9}$	N/m^2	$5.5149\cdot10^{8}$	N/m^2
Thermal expansion coefficient/ *Coeficient de dezvoltare termică*	$1.5\ \cdot10^{-5}$	/Kelvin	$9\cdot10^{-6}$	/Kelvin	$1.08\cdot10^{-5}$	/Kelvin
Thermal conductivity *Conductivitate termică*	24	W/(m.K)	6.7	W/(m.K)	1.4949	W/(m.K)
Specific heat *Căldură specifică*	590	J/(kg.K)	586.04	J/(kg.K)	877.96	J/(kg.K)
Hardening factor (0-1; 0=isotropic; 1=kinematic) *Factor de ecruisare (0-1; 0=isotropic; 1=kinematic)*			0.85			

Table 4

Material constants for the two types of bone/ *Constantele de material pentru osul trabecular şi osul cortical*

	Trabecular bone *Os trabecular*		Cortical bone *Os cortical*	
Constant name/ *Constantele*	**Value** ***Valoare***	**Unit** ***Unitate***	**Value** ***Valoare***	**Unit** ***Unitate***
Elastic modulus *Modulul elastic*	$1.8\cdot10^{8}$	N/m^2	$1.8\cdot10^{10}$	N/m^2
Poisson coefficient *Coeficient Poisson*	0.3		0.25	
Mass density/ *Densitate*	1500	kg/m^3	2000	kg/m^3
Tensile strength *Rezistență la tracțiune*	$2\cdot10^{7}$	N/m^2	$1.5\cdot10^{8}$	N/m^2
Yield strength *Limită de plasticitate*	$1.8e\cdot10^{7}$	N/m^2	$1.3\cdot10^{8}$	N/m^2

For the FEM models of the present study, the lateral sides and the bottom of bone are considered fixed (node displacements are blocked on those faces in all directions).

Establishing the loadings of the EF model, this is an important part of the study.

The loads that simulate the mastication forces generate stress concentration to be assessed and therefore should be considered an appropriate risk [7]. The size of the masticatory force can be variable depending on age, sex, edentulous, parafunctional habits and may vary even at the same pacient from anterior to posterior between 50-400 N [15].

3.Results

Since in implantology various types of DI are used, depending on the particular anatomical implant environment, we intend to investigate more models based on the most commonly used implants in implantology practice.

Thus, we performed some calculations using FEM program Cosmos, and in what follows a selection of results is presented. It is considered static loads, applied on the abutment/crown either by vertical forces of 80N, 120N, 140N, 160N or simultaneously by vertical forces in the same range and horizontal forces of 80N. For instance, the horizontal forces may simulate the lingual forces.

3.1. The determination of safety factor

The program CosmosWorks has a section for the determination of the distribution of safety factors calculated as a ratio of admissible limit values and FEM calculated values of stress. Stress admissible limit values are specific to each material and adopted in specific conditions described in the literature [2,3].

In the following let's present the safety factor distribution in the case of three suggestive calculations:

1. Case 1. The insertion in a portion of mandible of Rootform type implant. An implant prothesis is achieved by a ceramic crown, on which it is applied:
   Case 1a: vertical forces;
   Case 1b: vertical and horizontal forces simultaneously;
2. Case 2. The insertion in a portion of mandible of Rootform type implant. No implant prothesis is considered. Vertical forces of 120N are applied on the abutment;
3. Case 3. The insertion in a portion of mandible of Needle type implant. No implant prothesis is considered. Vertical forces of 120N are applied on the abutment.

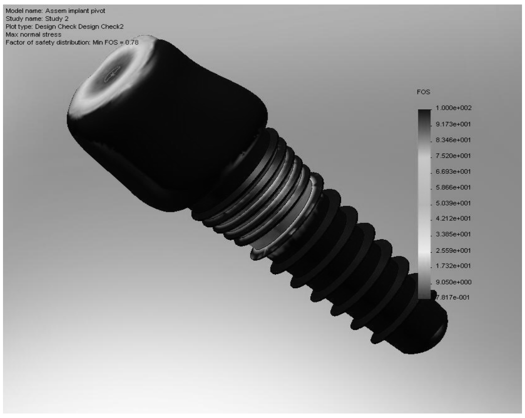
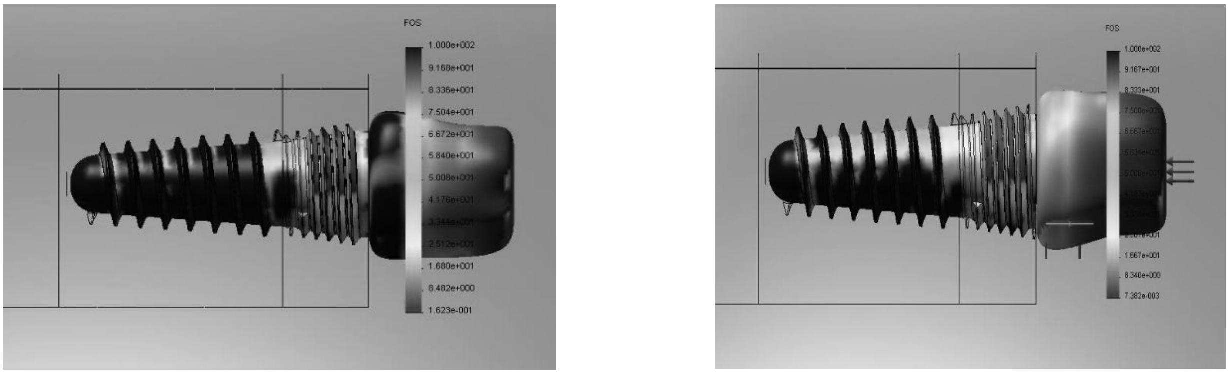


Fig.13 - Distribution of safety factor in implant Rootform and ceramic crown, axial load 120 N / *Distribuția factorului de siguranță în implant Rootform cu coroana ceramică, încărcare axială 120 N.*

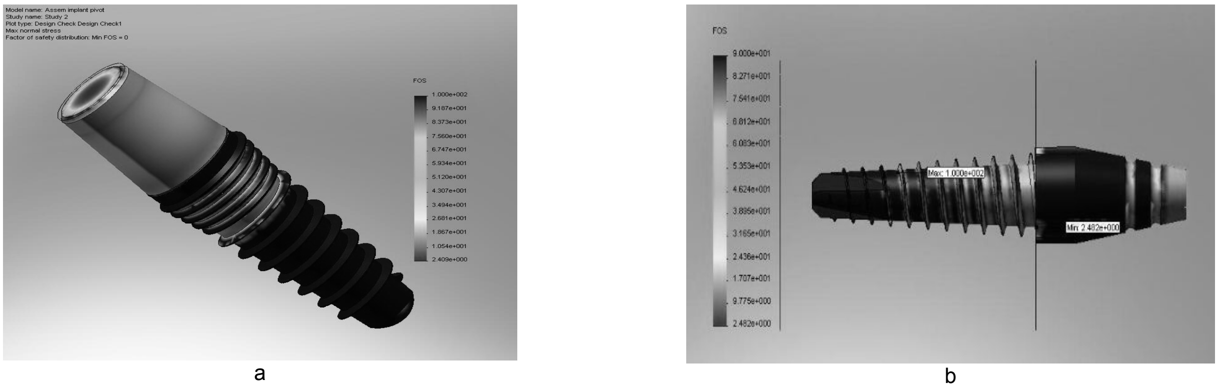
The values of safety factors are useful to indicate the critical areas having a low safety factor.

Figures 13, 14a and 14b represents the distribution of safety factor, in the case 1a (vertical force of 120N), case 1b (vertical force of 80N and horizontal force 80N simultaneously) and case 1b (vertical force of 120N and horizontal force 80N simultaneously), respectively.



a b

Fig.14 - Distribution of safety factor in implant Rootform and ceramic crown a. Axial load 80N, horizontal load 80N; b. Axial load 120N, horizontal load 80N / *Distribuția factorului de siguranță în implant Rootform cu coroana ceramică: a. încărcare axială 80N, încărcare orizontală 80N; b. încărcare axială 120N, încărcare orizontală 80N.*



a b

Fig. 15 - Safety factor distribution in implant: a. Rootform b.Needle / *Distribuția factorului de siguranță în implant: a. Rootform b. Denti Needle.*

Figures 15a and 15b presents the results concerning the safety factor in the case 2 and case 3, respectively. Thus, the same loading scenario of vertical force, such as in the case 1a is applied to a portion of mandible with an insertion of Rootform and Needle implant, with no crown.

We can compare the figures 13, 14a, 14b, 15a, 15b to analyze the distribution of the safety factor according the type of implant, to the loads intensity and type.

4. Discutions

From calculations, it appears that, regardless of the type of implant, the location of critical areas, those with lower values of the safety factor is the same, i.e. the implant neck. We also note that in this area, Rootform implant type has safety factor values of 2.4, close by the type of implant Needle, of 2.48, see Figures 15a and 15b.

Also, in Figure 16, the loads of combined vertical and horizontal forces (dashed line in figure 16) determines the critical areas more pronounced i.e. small safety factor, even 0003, for axial load 160N, 80N horizontal load versus axial loads of 160N with a safety factor of 0.4 (continuous line in figure 16).

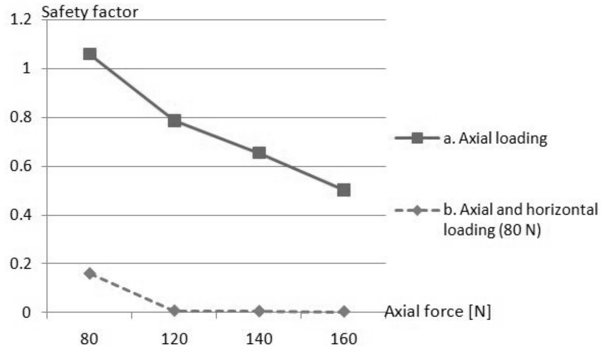


Fig. 16 - Minimum safety factor in implant Rootform and crown vs applied axial force / *Factorul de siguranță minim în implant Rootform cu coroană vs forța axială aplicată.*

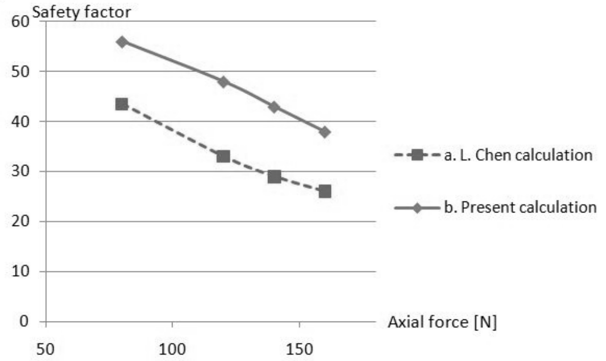


Fig. 17 - Safety factor in Rootform implant body with crown vs applied axial force / Comparația valorilor factorul de siguranță în corpul implantului Rootform cu coroană vs forța axială aplicată.

Regarding the safety factor estimation in implant body, let us present the results in Figure 17. In this figure, the results obtained in this study (solid line) are compared with results of a similar study by Chen in [15]. The differences are not significant, but they are due to the fact that in the present study, the bone-implant assembly is modeled very accurately and the loads simulation of mastication is done by applying vertical and horizontal forces, and in Chen's study, the whole bone-implant structure is modeled coarser, without involving all fine structural details, but mastication simulation is done by dynamic loads.

FEM analyzes are complementary to clinical trials and they simulate clinical situations. The results of FEM analysis cannot be implemented directly in clinical situations, but one can design a model to simulate a real situation as well as possible.

Simplifications and assumptions adopted are however some limitations of FEM studies. The FEM analysis should be interpreted with care. In most cases the numerical studies of oral implantology, for example, isotropic materials are used, but not orthotropic, or anisotropic, as would be more plausible.

In fact, the FEM model is a static state at a time of load application but not an actual clinical situation, in which the structure loading is rather dynamic and cyclical.

The numerical results obtained should be more precise and rigorous if the materials would be considered anisotropic and inhomogeneous, but they would lead to more complex mathematical calculations on one hand, but even more difficult, they would require complicated laboratory experiments to determine the material constants for living tissue type materials.

Another limitation of the FEM studies is the use, for instance, of Von Mises plasticity criterion, which in engineering is used rather for ductile material, as steel and aluminum.

However, new developments in computing and modeling techniques make the FEM a reliable and accurate approach in dealing with biomechanical applications [16].

FEM analysis results confirm conceptually that the materials interface with different modulus of elasticity represents a weakness of rehabilitation systems. Rehabilitation with materials having a modulus of elasticity similar to the tooth can save and strengthen the remaining tooth structure.

Combining fatigue laboratory tests with FEM analysis can eliminate or at least minimize the experimental limitations by the correlation of fracture by fatigue with the stress, instead of a specific test configuration [17].

Although, usually we use advanced computing technology to obtain numerical results, there are many factors that affect the clinical features such as the macro and micro design, material properties, loading conditions, boundary conditions [11].

Numerical models need to be run using many scenarios, e.g. for different values of loads, for various plausible zones of their applications, for various types of material, etc.

All numerical calculations require a correlation of results with preclinical data and clinical studies and a thorough validation.This is achieved by: comparing the results obtained for the same load scenario, boundary conditions using several numerical codes or standard EF programs; comparing results with actual clinical situations; comparing with available laboratory tests.

In order to validate thenumerical results we considered several discretisation variants as far as a smoothness that does not lead to solutions errors bigger than 3.5%.

5. Conclusions

This article is dedicated to the study of some types of biomaterials used in the manufacture of dental implants systems. The FEM study consists in providing some accurate geometric models of various types of DI, with which the distribution of safety factor is calculated for implant and the surrounding bone, obtaining therefore an evaluation of risk factors from a biomechanical point of view.

The originality of this study include, on one hand the development of some highly accurate modeling of different types of implant used in implantology, which represents exactly the real implants. Also, simulation of masticatory forces is done by applying axial and horizontal force load type.

Calculations show that the load type has a greater impact on critical areas than intensity. But regardless of the type of implants studied, critical areas develops around the neck of the implant and surrounding bone.

Studies using FEM has some advantages over clinical trials, pre-clinical or in vitro studies. Firstly, the patients are not in any way affected by the application of new materials and new treatment modalities that have not been previously tested.

In the field of biomaterials, FEM is an important tool as it avoids the need for a traditional specimen using instead a mathematical model that eliminates the need for a large number of teeth.

FEM analyzes are useful for design preparation, it indicates optimal materials or a combination of them to be used in various load conditions to reduce material consumption and/or avoiding their breaking in clinical practice.

.

In conclusion, the FEM analyzes are useful both to DI designers and clinicians and we believe that in the future they will develop even more, being accessible and available to a large number of clinicians.

Acknoledgements

I want to bring my warm thanks to Prof. Dr. E. Avram for unconditional support for the realization of the geometrical model and using FEM calculations. Both wish to thank to Prof. Dr. V. Năstăsescu for high professionalism and expertise regarding the use of FEM.

REFERENCES

1. B.C. Muddugangadhar, G.S. Amarnath, S. Tripathi, D. Suchismita, and M.S. Divya, Biomaterials for Dental Implants: An overview, International Journal of Oral Implantology and Clinical Research, 2011, **2**(1), 13.
2. E. Avram, and V. Năstăsescu, Application of FEM in fluid mechanics, Military Technical Academy Press, Bucharest, 1995.
3. V. Năstăsescu, FEM, Military Technical Academy Press, Bucharest, 1995.
4. O. C. Zienkiewicz, and R.L. Taylor, The Finite Element Method. 5th ed. Oxford, England, Butterworth-Heinemann,2000, **1**,1..
5. I. Roateşi , Implant modeling by finite element method, Romanian Journal of Stomatology, 2014, LX (4), 211.
6. I. Roateşi , Assessement of stress and strain state in implant and surrounding bone by finite element method, 2014, LX (4), 227.
7. X . Ding, X.H. Zhu, S.H. Liao, X.H. Zhang, and H. Chen, Implant-bone interfaces stress distribution in immediately loaded implants of different diameters: a three-dimensional finite element analysis, J Prosthodont, 2009, **18**, 393.
8. M.I. El-Anwar, and M.M. El-Zawahry, A three dimensional finite element study on dental implant design, Journal of Genetic Engineering and Biotechnology, 2011, **9**, 77.
9. F.J. de Cos Juez, F. S. Lasheras, P.J. G. Nieto, and A. Álvarez-Arenal, Non-linear numerical analysis of a double-threaded titanium alloy dental implant by FEM, Applied Mathematics and Computation, 2008, **206**, 952.
10. J.C. Vanegas-Acosta, P. N.S. Landinez, D.A. Garzón-Alvarado, and R.M.C.Casale, A finite element method approach for the mechanobiological modeling of the osseointegration of a dental implant. Comput Methods Programs Biomed, 2011, **101**(3), 297.
11. L. Haiyan, L. Jianying, Z. Zhenmin, and A.S.L. Fok, Fracture simulation of restored teeth using a continuum damage mechanics failure model, Dental Materials, 2011, **27**,125.
12. S.M.M. Rezaei, H. Heidarifar, F.F. Arezodar, A. Azary, and S. Mokhtarykhoee, Influence of Connector Width on the Stress Distribution of Posterior Bridges under Loading, Journal of Dentistry, 2011, **8**, 67.
13. * * * Cosmos Program Documentation.
14. * * * SolidWorks Program Documentation.
15. L. Chen, Finite element analysis of stress on the implant-bone interface of dental implant with different structures, ch. 3, Finite Element Analysis – New Trends and Developments, p. 55-76, INTECH, 2012.
16. J.Yanga, and H.J. Xiang, A three-dimensional finite element study on the biomechanical behavior of an FGBM dental implant in surrounding bone, Journal of Biomechanics, 2007, **40**, 2377..
17. O. Kayabasi, E. Yuzbasıoglu, and F. Erzincanlı, Static, dynamic and fatigue behaviors of dental implant using finite element method, Advances in Engineering Software, 2006, **37**, 649.

****