Romanian Biotechnological Letters *Vol. x, No. x,*

ORIGINAL PAPER

FEM analysis of masticatory induced stresses over surrounding tissues of dental implant

Received for publication, January, 6, 2018
Accepted, May, 12, 2018

ANDREEA ANGELA ȘTEȚIU[1], MIRCEA ȘTEȚIU[2], MIHAI BURLIBAȘA[3]*, VIOREL ȘTEFAN PERIEANU[3], GABRIELA TĂNASE[3], NARCIS MARCOV[3], OANA-CELLA ANDREI[3], RADU COSTEA[3], ELENA-CRISTINA MARCOV[3], DANA CRISTINA BODNAR[3], TRAIAN BODNAR[3], MAGDALENA NATALIA DINA[3], CONSTANTIN DĂGUCI[4], LUMINIȚA DĂGUCI[4], CORINA MARILENA CRISTACHE[3], CAMELIA IONESCU[3], LUMINIȚA OANCEA[3], MĂDĂLINA VIOLETA PERIEANU[3], CLAUDIA CAMELIA BURCEA[3], COSMIN MEDAR[3], CARMEN DANIELA DOMNARIU[1], ILEANA IONESCU[3]
[1]Faculty of Medicine, „Lucian Blaga" University, Sibiu, Romania, [2]Faculty of Engineering, „Lucian Blaga" University, Sibiu, Romania, [3]University of Medicine and Pharmacy „Carol Davila" Bucharest, Romania, [4]University of Medicine and Pharmacy of Craiova, Romania, [3]Corresponding author: Mihai Burlibașa, Faculty of Dentistry, No 19 Calea Plevnei, sector 5, Bucharest, Romania, e-mail: mburlibasa@gmail.com, phone: ++0723472632*

Abstract

In this paper are highlighted the stresses that appear in the peri-implantation area through the forces induced by the masticatory process. The analysis consists of computer modeling of the dental implant and simulation of the load with a distributed force. It is considered a variable direction loading related to the axial insertion of the dental implant, thus simulating an important masticatory process. This type of load reveals stresses developed in the periimplantary bone area, the risk areas where the stresses can cause the appearance of hard tissue damage near the implant and its failure. Increases in stresses up to 159% in the ruminant type of chewing compared with the cutter type are highlighted. An aggravating factor that can lead to implant failure is bruxism that tangentially stresses the implant leading to the concentration of forces at the abutment-implant junction. Conclusions regarding the decrease of stresses at bone-implant junction with up to 23% by a corresponding increase of the implant diameter by 25% are also issued.

Keywords: finite element method, dentistry, teeth modeling, dental implants, mastication.

1. Introduction

Although Oral Implantology is a relatively a new specialization, statistics show that the number of requests for implant supported prosthetic treatments has increased progressively, partly because of the refusal to treat Kennedy class 1 and 2 edentations with removable prostheses. Another implant option is found in patients with Class III and IV Kennedy edentations that do not accept interventions on teeth limiting edentulous breaches to complete a fixed dental prosthesis. These includes patients with single tooth edentations from visible areas (frontal and frontolateral) that can be fast treated with a dental implant with immediate or delayed loading with an aesthetic crown. Thus, due to the advantages offered by the dental implant for all types of edentation it becomes more and more demanded by the patients.

Among the problems that oral implantology has to deal with, beside those caused by the material the dental implant is made of so that it is supported and integrated by the bone tissue, there are the problems related to the shape and dimensions of the various systems that have to withstand in time the masticatory stresses (M. ȘTEȚIU & al [1], A. MERDJI & al

[2], T. HARALDSON & al [3], A. FRĂŢILĂ & al [4]). The dental implant is the physical support on which the dentures are fixed and through which the occlusal pressures developed in the mastication process are transmitted. Between the prostheses and the implant support that is inserted into the bone tissue, morphofunctional correlation should be created and achieved by balanced pressure distribution. Theoretically, an action of vertically oriented forces with equal and simultaneous distribution at the occlusal surfaces of implant supported prostheses must be achieved. Practically, through his own masticatory engrams, the patient develops variable forces and strains that can lead to the loss of periimplantary bone support and total or partial destruction of both the implant and the over denture (M. ŞTEŢIU & al [1], T. Haraldson & al [3], NASA [5]). Consequently, a deep analysis of the masticatory model, of the forces direction for a better predictability of the resistance of implant supported prostheses in time is required (M. ŞTEŢIU & al [1], M. BURLIBAŞA & al [6]). The morphological and clinical particularity of edentation and prosthetic areas are an important factor for diversification of dental implant types, especially those with screw shape, so that there are currently a multitude of such products accompanied by appropriate leaflets and tools. Choosing the type of implant in terms of shape, size, and material is established in the mean time with the treatment plan.

One of the problems that arise is the time resistance of prosthetic restoration and bone stress resistance under repeated action of the forces in the masticatory cycle, especially as it is known that a repetitive action of small amplitudes at tissues level can lead to necrosis of the adjacent tissue and the loss of the dental implant. We intend to highlight the dental implant risk areas through the finite element analysis of the 3D implant model.

2. Materials and Methods

Use of FEM in the analysis of periimplantation tissue stress

The finite element method (FEM) allows finding approximate solutions of partial differential equations and of integral equations as well. The solution approach is based either on eliminating the differential equation completely, or rendering the partial differential equations into an approximating system of ordinary differential equations, which are then numerically integrated (M. ŞTEŢIU & al [1], A. Merdji & al [2], A. FRĂŢILĂ & al [4], M. BURLIBAŞA & al [6], V. OLEKSIK & al [7], www.ansys.com [8], www.3ds.com [9], G. ODIN [10], Y. WEIJUN & al [11]).

In engineering this allows to search static analysis, dynamic or own values cases. For complicated cases one of them can be used to simplify the model and to solve differential equations on this simplified model (M. BURLIBAŞA & al [6], P. LI & al [12], V. LERTCHIRAKARN & al [13], T. ACHOUR & al [14], A. GERAMY & al [15]).

In the use of FEM we go through three stages: Pre-processing (MODELING), Processing (NUMERICAL CALCULATION) and Post-processing (VISUALIZATION AND ANALYSIS OF RESULTS).

Pre-processing (modeling) involves the finite element distribution of the mathematical model, made in software dedicated to this purpose (Solid Works, Catia, ProEngineer, Solid Edge, Unigraphics) and numerical modeling.

At this stage, special procedures are followed for preparing the geometry of the model, taking into account the field of use, materials, execution technology, physical phenomenon of stress and structural analysis (www.ansys.com [8], V. LERTCHIRAKARN & al [13], T. ACHOUR & al [14], B. GALER [16], A.A. ŞTEŢIU & al [17], T. VARADY &

al [18]). **Figure no. 1** shows the loading scheme for the masticatory forces. **Figure no. 2** shows the distribution of unitary efforts on the bone.

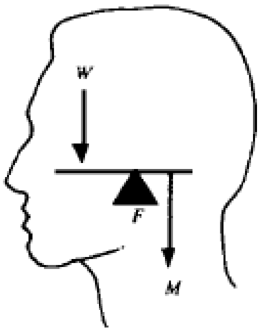


Figure no. 1. Forces and Supports - First-degree lever (NASA [19]).

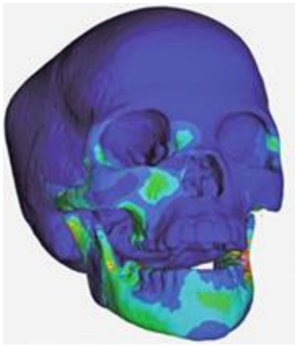
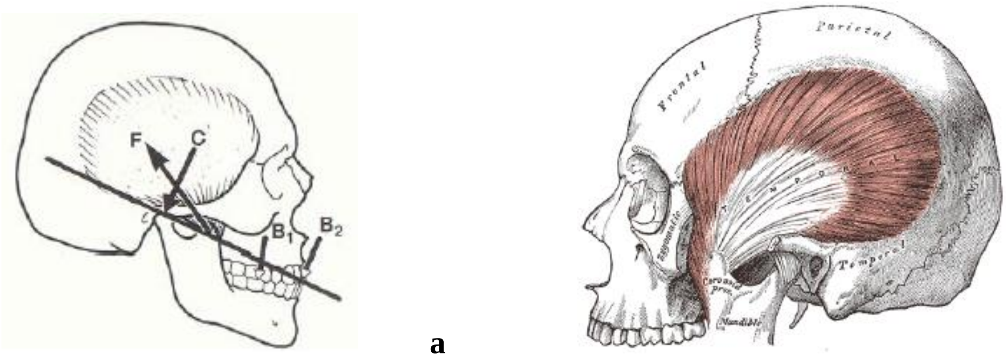


Figure no. 2. Distribution of unitary effort on bone (M. BURLIBAŞA & al. [6], NASA [19]).

The main jaw muscles, which will develop the masticatory force, are the temporal muscles pairs and masseters. In **Figure no. 3** we present the action of temporal muscle pairs (a) and masseters (b).



a

b

Figure no. 3. The force vector (B. GALLER [16]): a) temporal muscles; b) masseter muscles.

The finite element modeling is done at this stage and leads in obtaing elementary cells in the geometry content of a piece, the most often used being trihedral three-dimensional elements (with equilateral triangular faces), pentaedra (with equilateral triangular faces or squares) or hexahedron (cubic). One of the conditions that must be respected is that the geometry of the basic physical object must be respected, keeping the form and volume criteria. Another aspect is the attribution of physico-mechanical properties of model components. In particular, we have objects such as dental implant, compact bone and trabecular bone; as well as the composite and metallic materials from which overdenture is created and as specific loads are those given by the masticatory forces.

In this study we used two screw type cylindrical implants made of titanium. For the study to be relevant, we chose three implants of the same length but with different diameters (Ø 3.6, Ø 4.0, and Ø 4.5).

We modeled three-dimensionally and made the implant model mesh as in Figure no. 4 using professional software such as Solid Works, Catia, ProEngineer, Solid Edge, Unigraphics. For studying the mechanical behaviour of any structures the main goal is to obtain an accurate geometric model. The results of this phase are geometric models in one of formats such as: IGES, VDA, STL, DXF, OBJ, VRML and the most accurate CAD format for exporting a 3D model – ISO G Code.

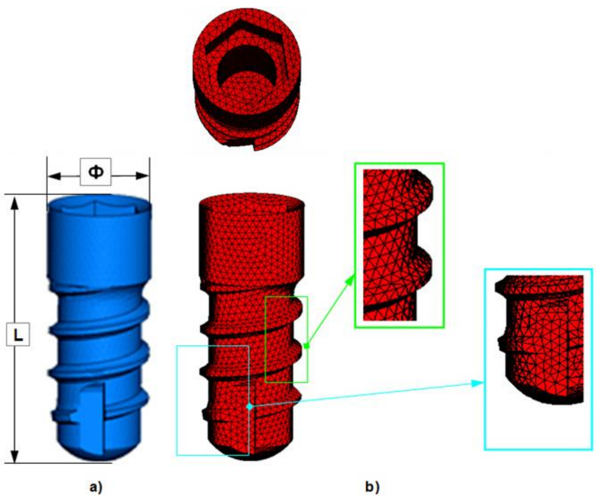


Figure no. 4. 3D model of the type 2 implant (a) 3D model; (b) 3D mesh model.

Figure no. 5 shows the interface nodes and mandible bone interface elements in which a type 2 screw implant was inserted. Since the Finite Element Method is quite representative regarding the value of surface and volume unitary effort, the mandible was dimensionally approximated with a prism of the following dimensions: length - L, thickness - G and height – H: L= 5 x $\Phi_{implant}$; G = 2 x $\Phi_{implant}$; H= 1.5 x $L_{implant}$. This approximation has been made because we will study the mandible effort only at the bone-implant interface.

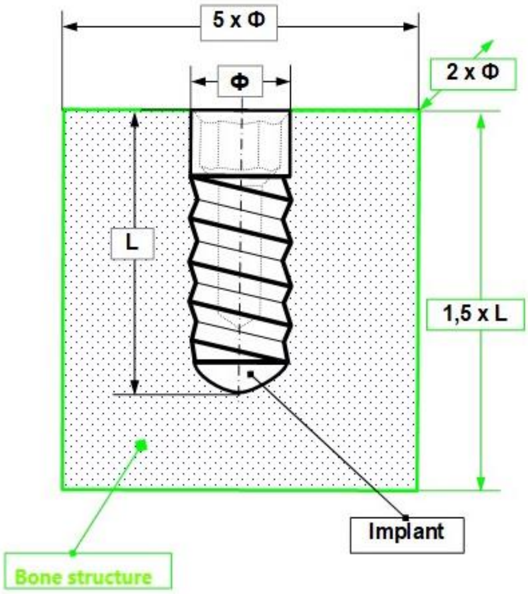


Figure no. 5. 3D model of the studied periimplantation area.

The bone-implant contact or bone-implant interface is actually a surface and will be modeled with special elements that will faithfully replicate the tension state at the bone - implant interface (**Figure no. 6, 7, 8**).

As a particular feature, modeling of the implant-bone interface, which is a direct contact between the two parts, can be done in two ways: with common nodes or intermediate interconnecting elements, **Figure no. 6.** The connection between the implant and the bone is made through special items element-element (surface-surface) so that the connection is modeled as realistic as possible, **Figure no. 7.**

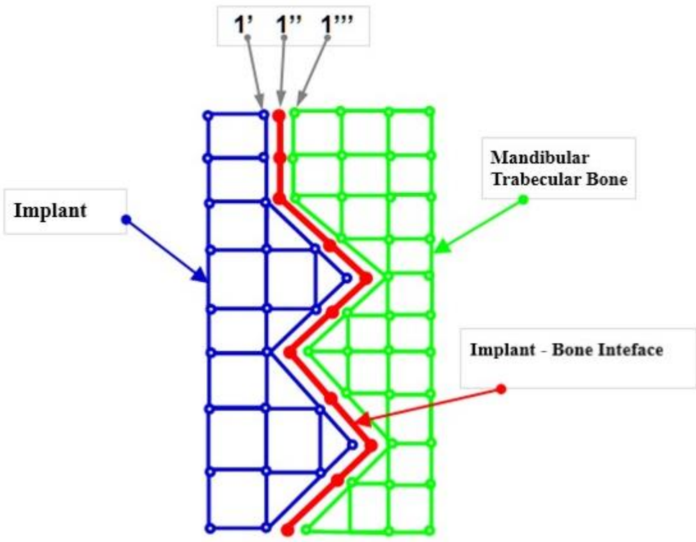
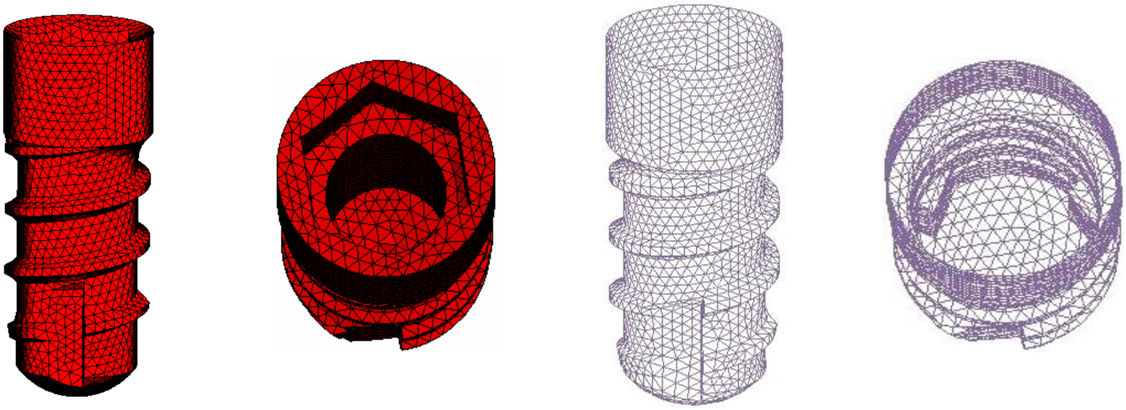


Figure no. 6. 3D-CAE modeling: Implant, mandible and implant-mandible interface.



Implant Implant-Bone Interface

Figure no. 7 – Implant and Implant-bone interface modeling.

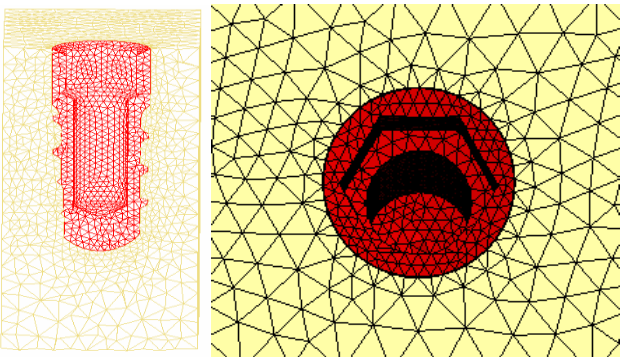


Figure no. 8. Section through implant-bone assembly and implant-bone interface mesh model.

Study models: Titanium dental implants are part of small size body type category and the minimum size of finite elements used for meshing will be 0.5 mm for implant and bone and less than 0.05 mm for implant-bone interface.

Depending on the proximity of the interface area the dimension of the bone elements follows a controlled progressivity, according to the quality criteria specific to this case. The implant and bone elements are of the three-dimensional type of tetrahedron.

The elements for the implant-bone interface are of the two-dimensional type, of the triangle type with special properties. Implant-bone interface elements have special conditions in common joint nodes of implant and bone.

The mandible consist of two different bone types structures: cortical on exterior and porous - trabecular on the interior.

The physico-mechanical properties of the mandibular bone introduced in all of our calculations are as follows (M. BURLIBAŞA & al [6], G. ODIN [10], Y. WEIJUN & al[11]):

- Density = 1,018 x $10e^3$ [kg/m^3]
- Young modulus
  - Cortical bone = 1,48x$10e^4$ [Mpa]
  - Trabecular high density bone = 1,58 x$10e^3$ [Mpa]
  - Trabecular low density bone = (2,31 – 48,0) x$10e^2$ [Mpa]
- Poisson Coefficient ν= 0,35 – 0,36

An important aspect of the modeling stage is the application of constrictions and load model with specific work forces. We will consider inserting the implant into the bone tissue so that the translations after the three axes will be completely canceled.

Processing (numerical calculation) involves analysis with professional software programs of the finite element calculation of the meshed model realized considering the model constrictions.

As an analyzing method adequate software such as ANSYS 11 or ABAQUS 6.14 can be used. The software permits to analyze the strain and stress state based on the discrete principle and modal analysis using the finite element method. So we determine the strain and stress state when loading the model in static regime. To analyze the stress distribution, the equivalent tension von Mises (σ_{ecv}) is used, and the program calculates it as quadratic average of normal stress at the base, middle or top of the finite elements (A. FRĂŢILĂ & al[4], M. BURLIBAŞA & al [6], V. OLEKSIK & al [7], www.ansys.com [8], www.3ds.com [9], Y. WEIJUN & al [11]).

Loading model stresses is made taking into account the active forces that raise the mandible and close the mouth. These forces are generated by lifting muscles, active in the mastication process (M. ŞTEŢIU & al [1], M. BURLIBAŞA & al [6], A.A. ŞTEŢIU & al [17], A.A. ŞTEŢIU & al [20], A.A. ŞTEŢIU & al [21]). Considering as loading the condition of a molar area implant (widespread distributed load) and taking into account that when loads are in the frontal area the stresses are higher and the risk of fracture increases considerably, the maximum loading force in the molar 1 area is around the maximum value of 390 N. On the other hand, the mean force, depending on the cross-sectional area of the muscle acting at a given moment, and the distance from the mandibular condylum, is 189 N at the same molar level (M. ŞTEŢIU & al [1], M. BURLIBAŞA & al [6], A.A. ŞTEŢIU & al [17], A.A. ŞTEŢIU & al [20], A.A. ŞTEŢIU & al [21]).

To determine the Von Mises equivalent unitary stress level in the bone area for the three implant types considered, we will work with the 50 N resultant force R (light mastication) acting in the vestibulo-lingual plane at an α angle of 0°, 30 °, 45 °, 60 ° and 90°.

3. Results and Discussions

Post-processing (visualization and analysis of results), taking into account the 50 N reference load, the load direction between 0 °-90 ° and the features presented above, for the areas surounding the three implant types, data from **Table 1** are obtained. **Figure no. 11** set out in detail stress diagram shows for the worst-case - a stress at an angle of 90° of the thinnest implant.

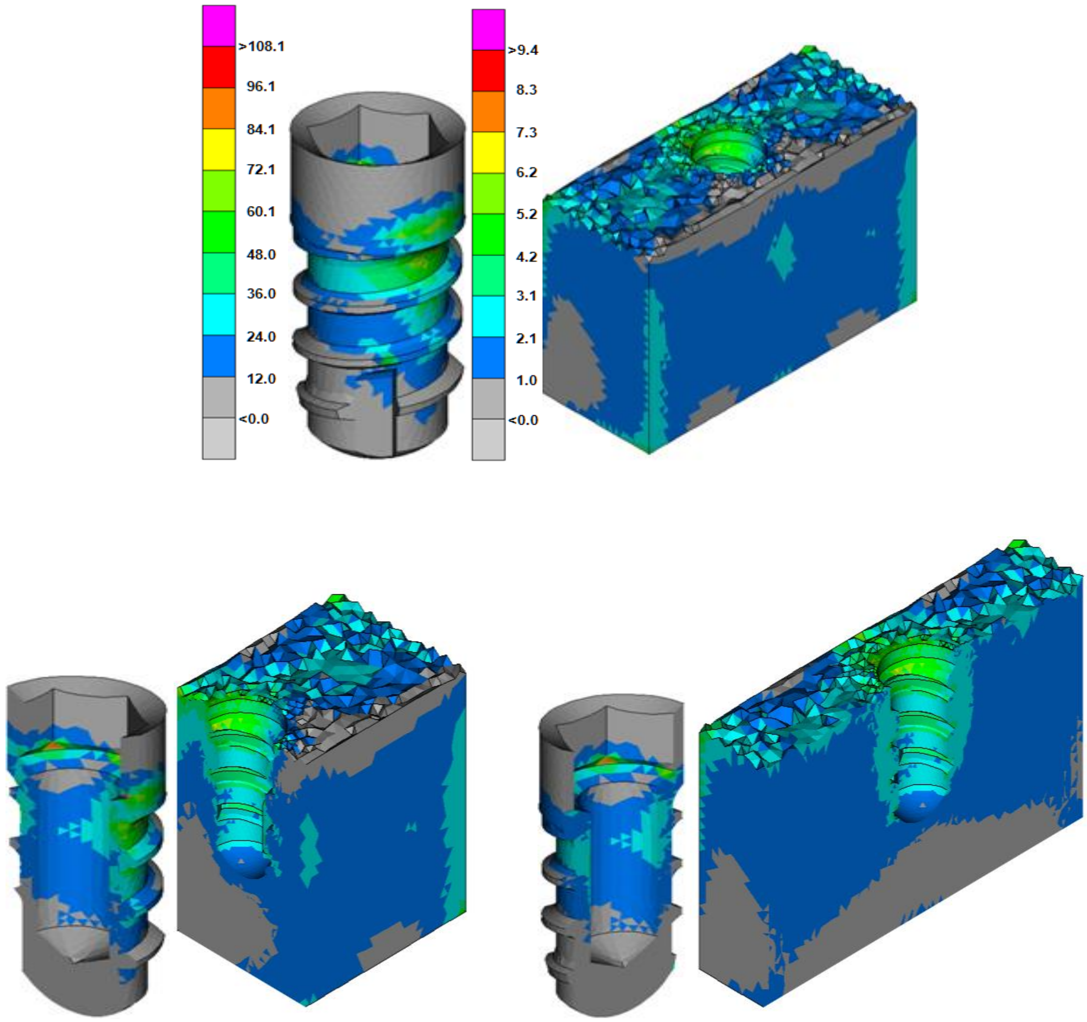
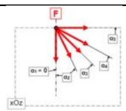
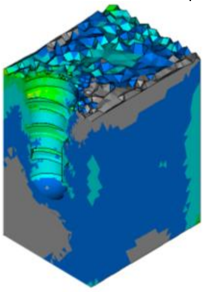
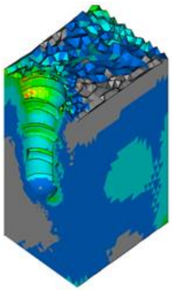
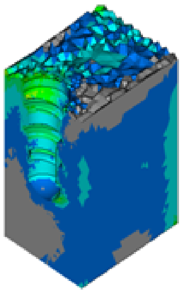
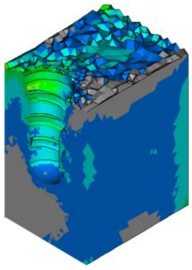
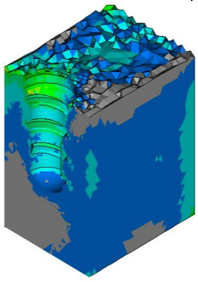
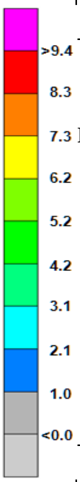
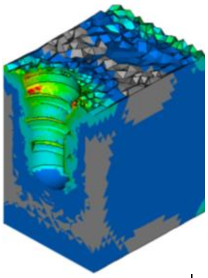
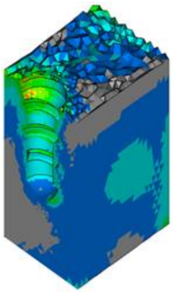
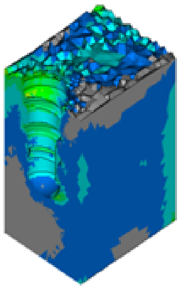
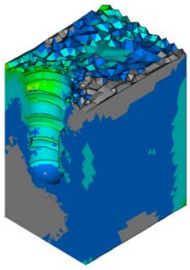
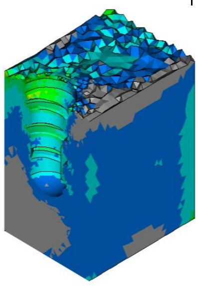
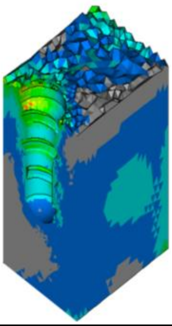
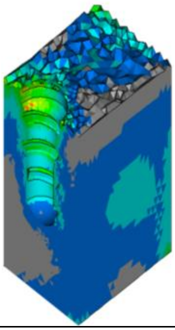
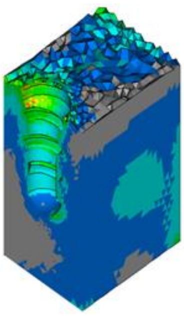
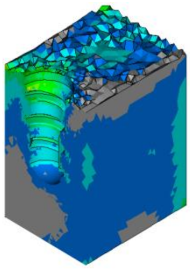
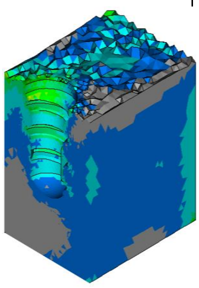


Figure no. 11. Von Mises equivalent stress distribution for Titanium implant, Φ = 3.6 mm; L = 9 mm, at an action of the masticatory force under an angle of 90º considered linguo-vestibular. Maximum Values σVM: Implant σVM = 108.1 [Mpa]; Mandible cVM = 9.4 [Mpa].

Table 1 summarizes the perimplantation bone stresses according to the type of implant and the angle of masticatory forces applied. For our research, we considered the alveolar bone resorption of 2 mm in the implant area.

Table 1. Results for implant loading with a force of 50 N at different angles in vestibulo-lingual plane (xOz)

Force 50 [N]	Stresses in bone	Angle of Force α [º] 

		0	30	45	60	90
	Von Mises maximum unitary stress [Mpa] For Type 1 Implant Φ = 3.6 mm; L = 9 mm	5,90 	6,70 	7,60 	8,90 	9,40 
	Von Mises maximum unitary stress [Mpa] For Type 2 Implant Φ = 4,0 mm; L= 9 mm	5,40 	6,10 	6,90 	8,10 	8,50 
	Von Mises maximum unitary stress [Mpa] For Type 3 Implant Φ = 4,5 mm; L= 9 mm	4,80 	5,50 	6,20 	7,20 	7,60 

4. Conclusions

The FEM model obtained manages to provide data on the state of strains and deformations that arise following a stress. The modeling being made after the implant blue print we can say it have been made without approximations, being a true copy of the geometric model.

The research allows the study of a wide variety of implants, space stresses and shapes by simply running the program with data and dimensions of the new study.

FEM research demonstrates the existence of a constant stress on bone-implant interface for all types of loadings. This endangers the resistance of the prosthetic work through changes in the periimplantar bone, and even the rejection of the implant can occur.

Another conclusion would be that connected to the masticatory type. Table 1 shows low Von Mises stresses in the case of vertical loading (chopper type) with an angle of 0º for

5. Acknowledgement: In this article, all the authors have equal contributions to the first author.

References

1. M. ŞTEŢIU, A. ŞTEŢIU. *Alternative analysis of masticatory forces*. Proceedings of the 4th International Conference on Manufacturing Science and Education – MSE 2009. Volume I, ISSN 1843-2522: 287-290, Editura Universităţii Lucian Blaga Sibiu, pp. 287-290 (2009).
2. A. MERDJI, R. MOOTANAH, B.A.B. BOUIADJRA, A. BENAISSA, L. AMINALLAH, E.B.O. CHIKH, S. MUKDADI. *Stress analysis in single molar tooth.* Materials Science and Engineering C 33: 691–698 (2013).
3. T. HARALDSON, U. KARLSSON, G.E. CARLSSON. *Bite force and oral function in complete denture wearers*. Journal of Oral Rehabilitation 6, Vol. 1, pp. 41–48 (1979).
4. A. FRĂŢILĂ, V. OLEKSIK, C. BOITOR, A. PASCU, B. PIRVU. *Numerical study about the strain analysis and the marginal design of dental indirect restorations*. Romanian Biotechnological Letters, Vol. 17, No. 4, pp. 7474-7482 (2012).
5. * * * NASA and Specialists. *Physics of the body muscle and forces.* 2010: 83 (2010).
6. M. BURLIBAŞA, A.A. ŞTEȚIU, D.G. MARINESCU, M. ŞTEŢIU, A. RĂDUCANU, A. TEMELCEA, N. MARCOV, V.Ș. PERIEANU, L. OANCEA, N. MĂRU, C. IONESCU, M. MALIȚA, M. DAVID, R. COSTEA, G. TĂNASE, L. ŞTEF, G. BOȚA, M.R. GLIGOR, C. DOMNARIU, B. PAVĂL, L.G. SFARGHIU, I. IONESCU. *Finite element method analysis of the stress induced upon the dental implant by the mastication process*. Romanian Biotechnological Letters, Vol. 22, No. 4, pp. 12706-12714 (2017).
7. V. OLEKSIK, A. PASCU. *Modern methods of study and research in mechanical engineering applied to medicine*. Proceedings of 5th Balkan Region Conference on Engineering and Business Education & 2nd International Conference on Engineering and Business Education, Sibiu, Romania (2009).
8. *Ansys.* [Online] [Cited: March 16, 2017.] http://www.ansys.com.
9. Abaqus Unified Fea. [Online] Dassault Systems. [Cited: 02 06, 2017.] http://3ds.com/products-services/simulia/products/abaqus/latest-release/.
10. G. ODIN. *These: Modelisation numerique de l'os mandibulaire appliquee a l'implantologie dentaire et maxilo-faciale*. L'Ecole Nationale Superieure des Mines de Paris, Directeur de these – Jean-Marc HAUDIN, Sutenue le 25 juin 2008, pp. 30; 37; 64; 65; 79 (2008).
11. Y. WEIJUN, J. XIAOHUI, S. PING, C. WEI. *Stress distribution in the mandible central incisor and periodontal ligament while opening the bite: A finite element analysis.* Scientific Publishers of India, Biomedical Research 23(3): 343-348, ISSN 0970-938X, pp. 344 (2012).
12. P. LI, J. MAO, Z. PENG. *Three-dimensional finite element analysis of the mechanical stress on root from orthodontic tooth movement by sliding mechanics*. J Huazhong Univ Sci Technolog Med Sci. 27, 745, 747 (2007).
13. V. LERTCHIRAKARN, J.E. PALAMARA, H.H. MESSER. *Finite element analysis and strain-gauge studies of vertical root fracture*. J Endod. 29, 529, 534 (2003).
14. T. ACHOUR, A. MERDJI, B. BACHIR BOUIADJRA, B. SERIER, N. DJEBBARAL. *Stress distribution in dental implant with elastomeric stress barrier.* Mater Design 32, 282, 290 (2011).
15. A. GERAMY, S. FAGHIHI. *Secondary trauma from occlusion: three-dimensional analysis using the finite element method.* Quintessence Int. 35, 835, 843 (2004).
16. B. GALER a.o. *Human jaw motion simulator.* Iris, Northeastern University, Department of Mechanical and Industrial Engineering, pp. 18-32 (November, 05, 2007).
17. A.A. ŞTEŢIU, M. OLEKSIK, V. ŞTEŢIU, R. PETRUSE, M. BURLIBAŞA, M. CERNUŞCĂ-MIŢARIU, I. IONESCU. *Dimensioning initial preparation for dental incisal reconstruction with biomaterials*. Romanian Biotechnological Letters, Vol. 19, No. 5, pp. 9792-9800 (2014).
18. T. VARADY, R. R. MARTIN, J. COX. *Reverse engineering of geometric models - An introduction.* Computer Aided Design. Vol. 29, No. 4, pp. 255-268 (1997).
19. NASA and Specialists. *Physics of the body muscle and forces*, pp. 83, (2010).
20. A.A. ŞTEŢIU, M. OLEKSIK, V. OLEKSIK, M. ŞTEŢIU, M. BURLIBAŞA. *Mechanical behavior of composite materials for dental obturations*. Romanian Biotechnological Letters, Vol. 18, No. 4, pp. 8528-8538 (2013).
21. A.A. ŞTEŢIU, M. OLEKSIK, V. OLEKSIK, R. PETRUSE, M. ŞTEŢIU. *The minimal burr dimensioning of teeth preparations to be restored with biomaterials*. Romanian Biotechnological Letters, Vol. 18, No. 3, 8381-8390 (2013).