ISSN print: 1224-5984
ISSN online: 2248-3942

Rom Biotechnol Lett. 2020; 25(4): 1802-1809
doi: 10.25083/rbl/25.4/1802.1809

Received for publication, October, 2, 2019
Accepted, January, 2, 2020

Original paper

Biomechanical Interest of Artificial Periodontal Ligament in Dental Implantology: A Finite Element Study

ALI BENAISSA[1], ALI MERDJI[1,2], IYAD MUSLIH AL-SARTAWI[3], MIRCEA STETIU[4,*], MSOMI VELAPHI[5], RAJSHREE HILLSTROM[6], SANDIPAN ROY[7], PERK LIN CHONG[8], MOHAMED EL-AMINE DJEGHLAL[9], BEL-ABBÈS BACHIR BOUIADJRA[2], ANDREEA ANGELA STETIU[4] , OSAMA MUKDADI[10]

[1]Faculty of Science and Technology, Mascara University, Algeria
[2]Laboratory of Mechanics and Materials Physics (LMPM), Mechanical Engineering Department, University of Sidi Bel-Abbes, Algeria
[3]Energy Services Center, Amman, Jordan
[4]Lucian Blaga University, Sibiu, Romania
[5]Faculty of Engineering and the Built Environmment (FEBE), Cape Peninsula University of Technology, Bellville, Cape Town, South Africa
[6]Department of Bioengineering, Tandon School of Engineering, New York University, USA
[7]Department of Mechanical Engineering, SRM Institute of Science and Technology, Chennai-603203, India
[8]Engineering Processes Research Group, School of Computing, Engineering & Digital Technologies, Teesside University, Middlesbrough, UK
[9]Department of Mechanical Engineering, National Polytechnic School, Algiers, Algeria
[10]Department of Mechanical and Aerospace Engineering, West Virginia University, Morgantown, WV 26506, USA

Abstract

Finite element method (FEM) is an important tool used in our days even in medicine was the relationship between the human body and artificial structure can be predicted. This work presents a numerical study performed with FEM of new dental implant system. A conventional dental implant system was redesigned and an artificial periodontal ligament was interposed between the implant and the alveolar bone. The aim was to attenuate the stress in the bone surrounding the implant. The new system was assessed and the interface stresses compared with the ones provoked by the conventional implant. In general, the novel dental implant provoked lower interface stresses due to the stress shielding effect of the artificial periodontal ligament.

Keywords Dental implant, periodontal ligament, stress, finite element method.

To cite this article: BENAISSA A, MERDJI A, AL-SARTAWI IM, STETIU M, VELAPHI M, HILLSTROM R, ROY S, CHONG PL, DJEGHLAL ME-A, BOUIADJRA B-AB, STETIU AA, MUKDADI O. Biomechanical Interest of Artificial Periodontal Ligament in Dental Implantology: A Finite Element Study. *Rom Biotechnol Lett*. 2020; 25(4): 1802-1809. DOI: 10.25083/rbl/25.4/1802.1809

✉ *Corresponding author: MIRCEA STETIU, Lucian Blaga University, Bd-ul Victoriei, Nr. 10, Sibiu, 550024, Tel.: 0269 218 165, Romania
E-mail: stetium@yahoo.com

Introduction

Dental implant has been increasingly used to recover the masticatory function of lost tooth. It has been well known that the success of dental implant is heavily dependent on initial stability and long-term osseointegration due to optimal stress distribution in the surrounding bones.

Stress and strain fields around osseointegrated dental implants are affected by a number of biomechanical factors, including the type of loading, material properties of the implant and the prosthesis, implant geometry, surface structure, quality and quantity of the surrounding bone, and the nature of the bone–implant interface (KOCA, 2005; ŞTEŢIU, 2019). As far as implant shape is concerned, design parameters that primarily affect load transfer characteristics (the stress/strain distributions in the bone) include implant diameter and the length of the bone–implant interface, as well as, in the case of threaded implants, thread pitch, shape, and depth. To increase the surface area for osseous integration, threaded implants are generally preferred to smooth cylindrical ones (PAPAVASILIOU, 1996). Depending on bone quality, surface treatments and thread geometry can significantly influence implant effectiveness, in terms of both initial stability and the biomechanical nature of the bone–implant interface after the healing process (COCHRAN, 1999).

The biomechanical behavior of dental implant is quite different from natural teeth. One of the major reasons is that for dental implants, there is a lack of function of periodontal ligament. That is because material of periodontal ligament is a soft tissue, and it could function as an intermediate cushion element which absorbs the impact force and uniformly transfers the occlusal forces into the surrounding bone. However, the bio-structure of dental implant is directly connected with bone. That would cause the non-uniform stress pattern at bone and might induce biomechanical overloading failures in implant and bone. This overloading would cause the micro damage accumulation at bone and results in primary marginal bone loss. Then the bacterial invasion might occur in the area of bone loss and cause serious progressive bone resorption (HANSSON, 2003). This insufficient bone support is dangerous for implant stability and might increase the risk of implant fracture and bone failure (SPIEKERMANN, 1995).

For this reason biomechanical optimization is an important objective in the design of dental implants several concepts have been developed, and many implant types are commercially available in different sizes, shapes, materials, and surfaces. To analyze the effectiveness and reliability of endosseous implants, revealing possible risks of implant failure, stress analysis of bone–implant mechanical interactions is important (GERAMY, 2004). The study for artificial periodontal ligament has become an important issue in this field. Thus, a new concept of coating an implant's surface with a natural polymer membrane was introduced in order to provide the viscoelastic characteristic of the periodontal ligament to implant system.

In recent years, the finite element method has been used to investigate the stress distribution within implant dentistry (COCHRAN, 1999). This method can be used as an ideal tool to investigate the functional responses of dental implants in different conditions. It allows the investigation of the relative merits of different parameters, shapes or designs as well as offering insight into the internal state of stress in components or materials within the implant or at the implant–bone interface (ISHGAKI, 2002). In this study, artificial ligament was coated on Bränemark type dental implant for replacing the role of intact periodontal ligament.

Material and Method

A. Geometrical Models

The modeling consists of using CAD software to create three-dimensional models representing the implant systems were based on Brånemark system and mandibular bone respectively. In this study two different types of implant system were compared (Figure 1):

- The conventional implant system is composed primarily of four parts: (a) the crown, (b) the framework, (c) the abutment screw, (d) the Abutment, (d) and (e) the implant.
- The new implant system is composed with the same parts of the conventional implant system and (f) the artificial ligament was interposed between the bone and the implant.

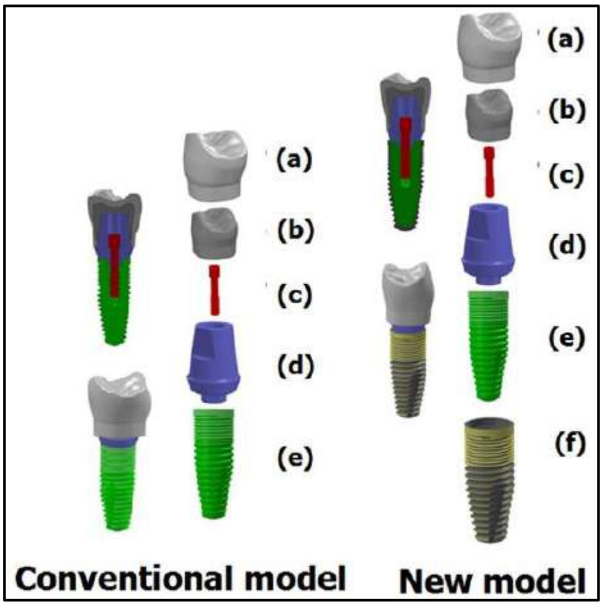

Figure 1. Components of the models

The mandibular bone the original 3D model of a mandibular bone section was constructed using computerized tomography (CT) scan technology (Fig. 2). The mandibular section was processed in Solidworks 3D (CAD, Software-2012), on which the final 3D solid model of the mandibular bone was created. The bone was modeled as a cancellous core surrounded by a cortical layer. The width and height of cortical bone model were 15.8 mm and 23.5 mm, respectively. The thickness of its upper part was 2 mm (Fig. 3).

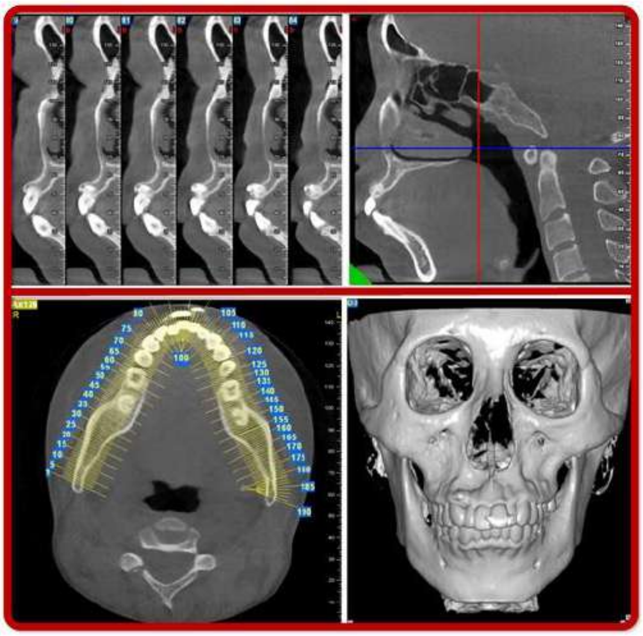

Figure 2. Computer Tomography (CT) scan of patient.

B. Material properties

The material properties adopted were specified in terms of Young's modulus, Poisson's ratio and density for the implant and all associated components (Table 1). All materials were assumed to exhibit linear, homogeneous elastic behavior (KAYABAŞ, 2006).

C. Boundary conditions

In order to define the boundary conditions, a 3D coordinate system was defined by three dynamic loads in the coronal–apical direction, lingual–buccal direction and mesial–distal direction.

For the boundary conditions, 3 zones were considered (Fig. 4):

- The inferior plane of the mandibular bone was defined as having zero displacement.
- The central surface in the occlusale face of the crown was subjected to a combined load of 17.1 N, 114.6 N and 23.4 N in a lingual–buccal, a coronal–apical, and a distal–mesial direction, respectively.

The other surfaces were treated as free surfaces, i.e. zero loads.

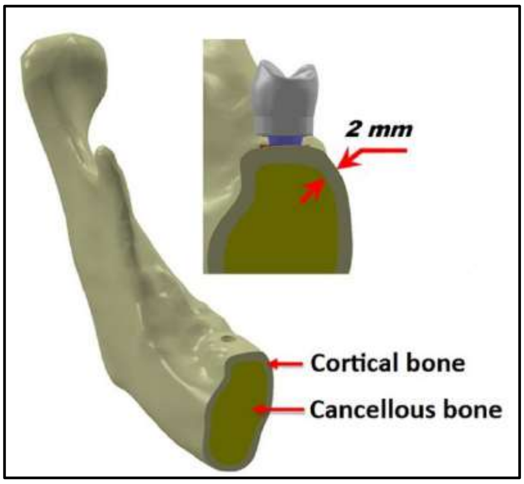

Figure 3. Components of mandibular bone

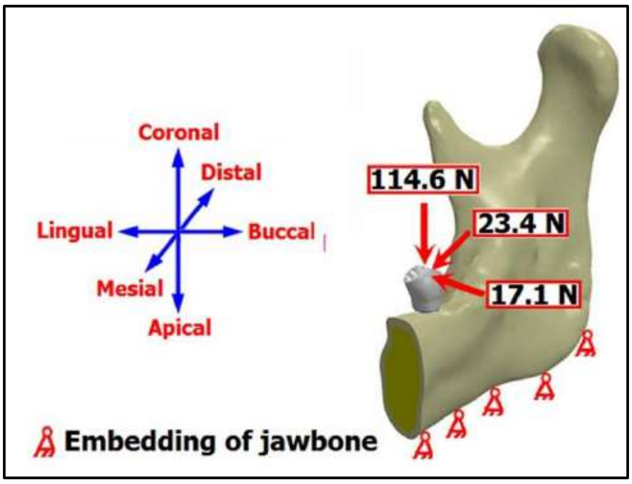

Figure 4. Boundary conditions

Table 1. Mechanical properties of investigated materials (KAYABAŞ, 2006)

Parts	Materials	Elastic modulus, E [Gpa]	Poisson's ratio	Density [kg/m^3]
Crown	Feldspathic porcelain	61.2	0.19	2300
Framework	Co–Cr alloy	218	0.33	8500
Abutment	Titanium	110	0.32	4428.8
Implant	Titanium	110	0.32	4428.8
Abutment screw	Titanium	110	0.32	4428.8
Mandibular bone	Cortical bone	$E_x = E_y = 11.5$ $E_z = 17$ $G_{xy} = 3.6$ $G_{xz} = G_{yz} = 3.3$	$\nu_{xy} = 0.48$ $\nu_{xz} = \nu_{yz} = 0.31$	1100
	Cancellous bone	3	0.29	270
Artificial ligament	Silicone	0.006	0.49	2220

For dynamic analysis, time dependent masticatory load is applied. Time history of the dynamic load components for 4 s is demonstrated in Figure 5. The solid model resulting from the intersection of implant and jaw bone represents the assumption of complete osseointegration, restricting any relative displacement between implant and bone.

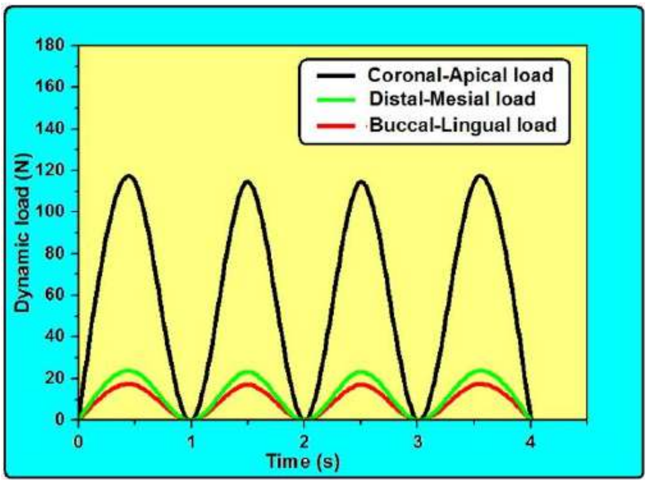

Figure 5. Dynamic loading in 4 s.

D. Finite Element Model

The mesh of the components is simplistic and consisted of linear tetrahedron elements with four nodes (Fig. 6). Since the interface of bone–implant experiences the largest deformations under load, it is necessary to mesh this boundary into small elements. The implant system and the bone were meshed with increasingly larger elements as the distance from the interface increases, with the size of elements in contact with the interface being defined by the elements of the boundary mesh.

Results

In this study, the distributions of the von Mises stress in the bone surrounding the implant were investigated. The von Mises stress is a scalar variable that is defined in terms of all the individual stress components and, therefore, is a good representative of the state of stresses. It has been extensively used in biomechanical studies of bone and dental prostheses (GENG, 2001).

The distributions of overall stress state for each component in our model were shown under effect of axial and horizontal loading in the coronal–apical, lingual–buccal and distal–mesial. A qualitative and quantitative analysis was performed, based on a progressive visual color scale, pre-defined by the software used, ranging from dark blue to red (Fig. 7). The maximum stress values in each component under different loading are shown in (Fig. 8).

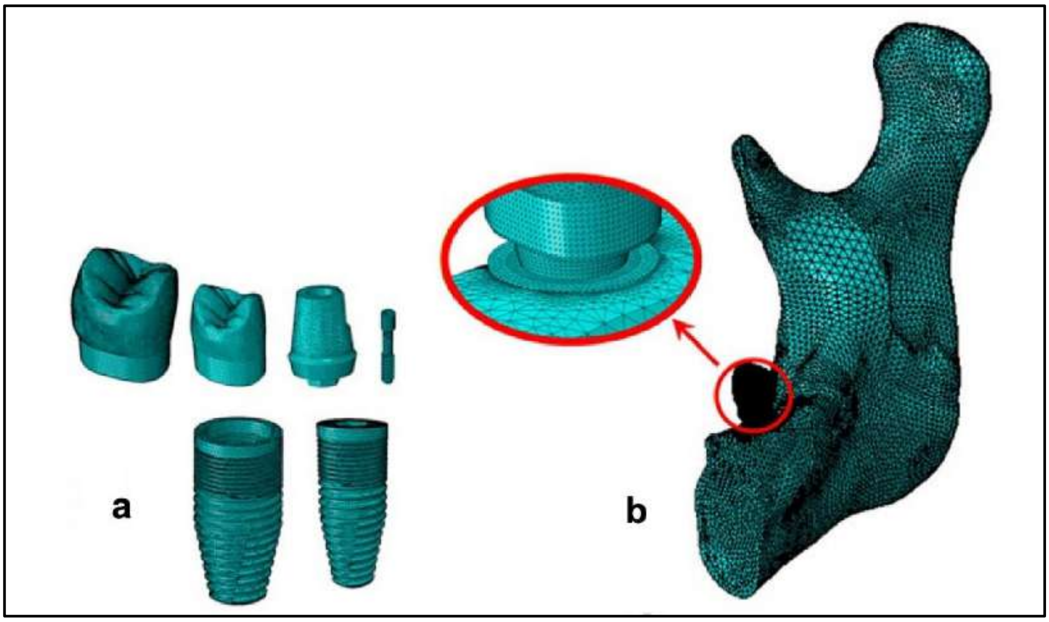

Figure 6. Mesh using linear tetrahedron elements of: (**a**) the parts of the implant system and (**b**) the mandibular bone and the final model.

In this section, the von Mises stresses were obtained from the analysis, allowing the consideration of maximum compressive and tensile stresses, as bone behavior under tension and compression is essentially different. Figure 9 shows points distributed along the implant–bone interface at a cervical, bucco-lingual and a pathmesio-distal section used to plot the von Mises stresses variation. Along the paths shown in the same figure, graphics were generated to make comparisons between both implant system geometries, displaying maximum and minimum von Mises stresses for both models under combined dynamic loads.

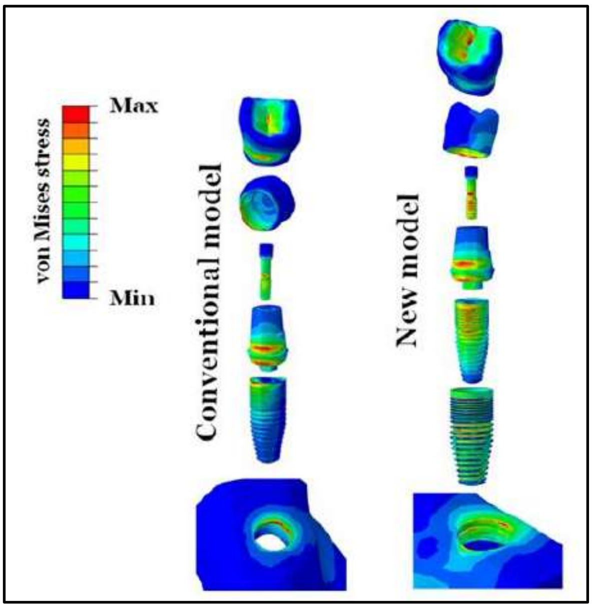

Figure 7. The distributions of overall stress ranging from dark blue to red.

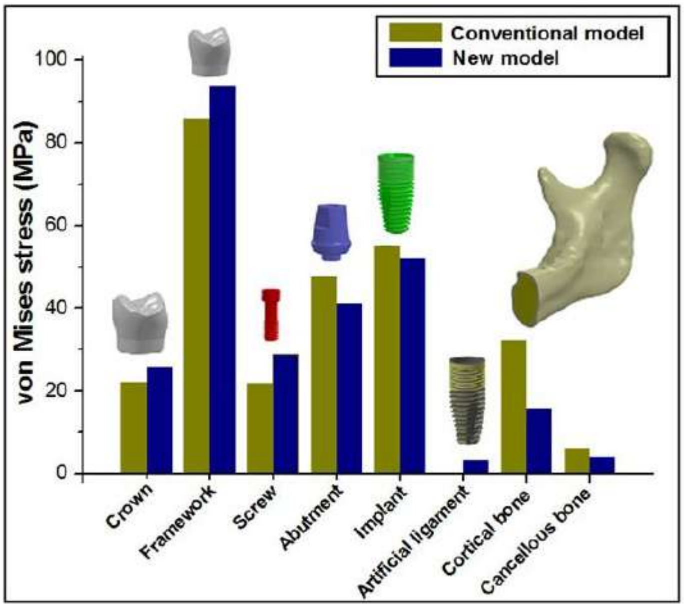

Figure 8. Histograms of comparison of von Mises stresses for each component in both models.

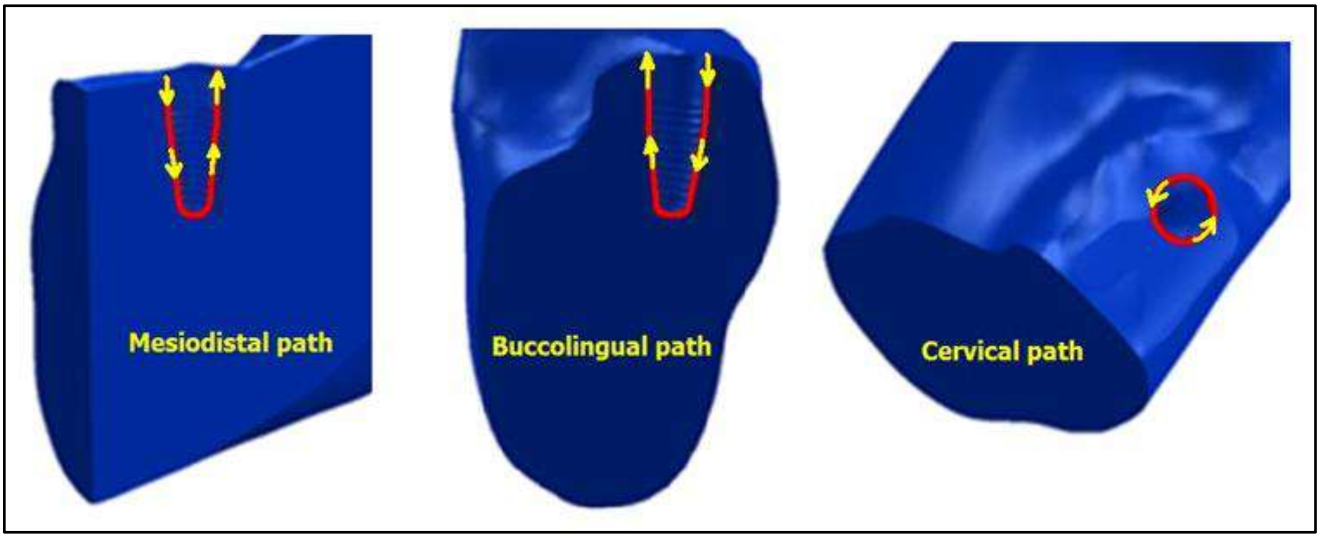

Figure 9. Different paths of the bone-implant interface used for stress distribution.

Figures 10, 11 and 12 present a comparison of von Mises stress distribution along the cross-section of both models for the three different types of loadings. The largest tensile stresses occurred in the cortical bone in one side loaded under the larger curvature region of the crown surface in the cervical area while the highest stresses occurred on the cervical line (Fig. 10).

The conventional implant under dynamic load was presented a high compressive peak stress concentration in one side of the cortical bone around the implant and a smooth distribution along the body of cancellous bone (Fig. 11 and 12). These stresses, decreased in the coronal–apical direction, and sudden a slight ascends on the curves shows the increase in stresses at the base of the implant on the same side as those in cortical bone (Fig. 13). For the new implant system with artificial ligament in mesiodistal direction path (Fig. 11), the stress distribution was qualitative similar with the conventional implant; however, there is a big difference in the cortical bone. A similar pattern occurred for buccolingual direction path (Fig. 12), although reaching different values.

In general, the curves show that the stress distribution at the interface in the bone of the model with an artificial ligament was lower than for the conventional model.

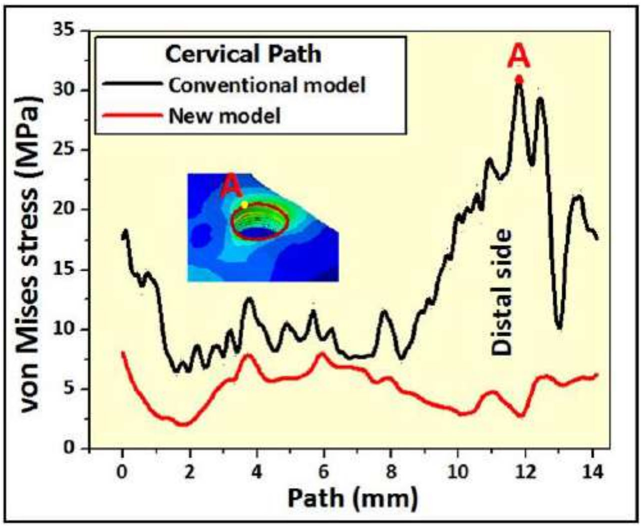

Figure 10. Comparison of stress distribution around bone/implant interface (Cervical path).

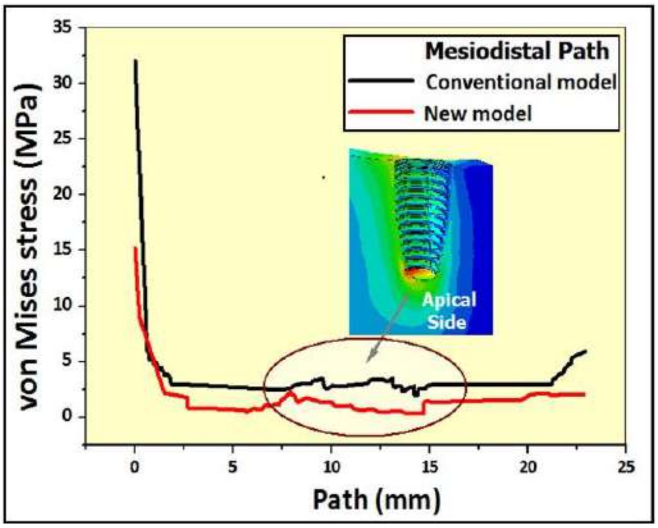

Figure 11. Comparison of stress distribution around bone/implant interface (Mesiodistal path).

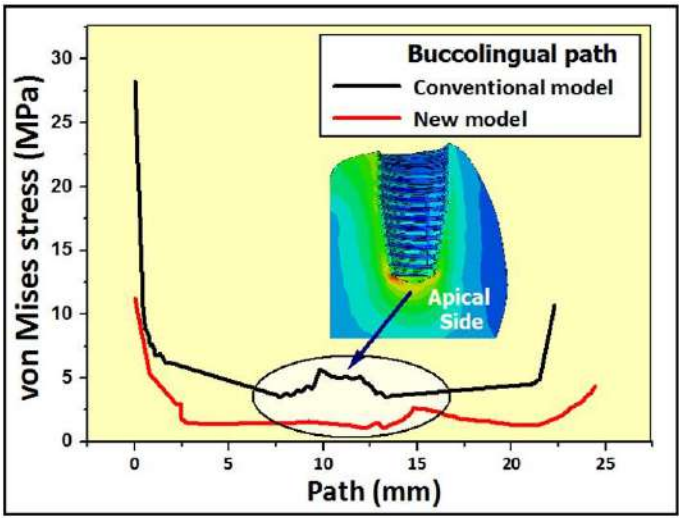

Figure 12. Comparison of stress distribution around bone/implant interface (Buccolingual path).

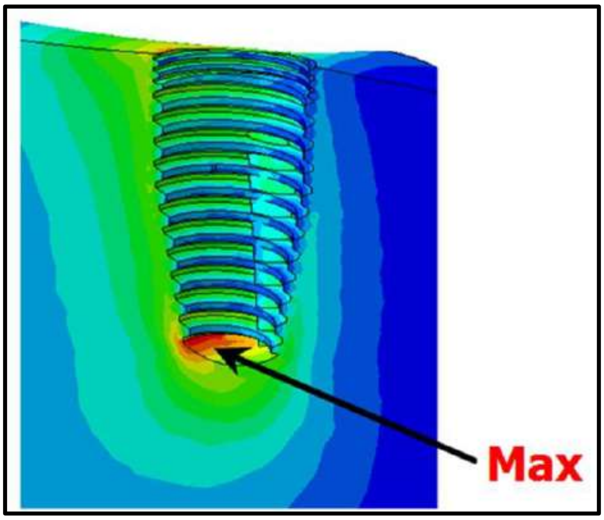

Figure 13. The stress contours of cancellous bone for buccal side.

Discussion

The aim of this study was to provide an analysis between two different geometric configurations of implant systems, to find the pure effect upon the bone stresses of prosthesis materials, to know the influence of the artificial ligament on the load transfer to the bone and to compare their biomechanical behavior. For this reason, it was assumed that all the parameters of both models were identical except the structural part of prosthetic design.

In both models, the extreme stresses in the mandibular bone occur in the layer of cortical bone adjacent to the neck of the implants. These were due to:

- The evidence of the surface area between the implant and the cortical bone is much smaller than the surface area between the implant and the cancellous bone. In addition, the cortical bone is more than ten times stiffer than the cancellous bone. These are the reasons due to the high stress increments were found in the cortical bone.
- The intimate contact at the cortical bone and implant interface; the loading applied to the implant is directly transmitted to the cortical bone.

This suggests that great importance is to be attached to the contact of the implant with the cortical layer of bone.

In a number of radiologic long-term studies, loaded implants showed typical bone loss around the implant neck (NATALI, 2006). This agrees well with the results of the present finite element study, in which the highest stress levels occurred in this very area. The cervical bone resorption always occurs to accommodate the reformation of a ‘biological width’. Preservation of peri-implant bone height depends on the magnitude and concentration of stress transmitted to the bone by the implant. There appears to be an optimal level of stress at which bone resorption is balanced by apposition. The minimum required load for avoidance of cortical bone loss appears to have been defined, but the upper limit of the physiological stress range has not yet been fully investigated.

In order to improve osseointegration, recent studies have focused on implant position, shape, and surface characteristics (COOPER, 1998; VAN STEENBERGHE, 1995; LUMBIKANONDA, 2001). Stress around implants may lead to bone resorption and implant loss (TADA, 2003; KITAMURA, 2005; VAN OOSTERWYCK, 1998; STETIU, 2019; BURLIBAŞA, 2017). Therefore, determining the stress distribution and intensity is important for understanding the process that leads to the loss of integration.

In the present study, stresses in the new implant were in general lower than in conventional geometry, indicating that stress-induced bone resorption should not be more critical in this geometry than in more usual conventional implants. This fact was expected, as the indication of use of such implant geometry comes not from the need to reduce stresses, but from occasional anatomic difficulties in the use of more traditional solutions.

In our model, the larger differences in peak stresses were for horizontal loading; this increase was also larger for the conventional implant. Interpretation of the numerical results should take into account that in this study, during mastication, the horizontal components of the loading are higher than the vertical components and in parafunction, vertical loads can be dominant, representing a specially critical situation.

Finite element analysis is based on mathematical calculations while living tissues are beyond the confines of set parameters and values since biology is not a computable entity. Consequently, finite element analysis should not be considered alone as a sole means of understanding the behavior of a geometrical structure in a given environment. Actual experimental techniques and clinical trials should follow finite element analysis to establish the true nature of the biologic system.

Conclusions

Stress analysis of two different geometries, a conventional model and a new model with artificial ligament, was performed using the finite element method, leading to the following conclusions:

- Both studied geometries presented quite similar qualitative stress distributions;
- Stresses in the new implant system with artificial ligament were in general lower than in the conventional implant;
- In both geometries stress concentration occurred at one side of the neck;
- High magnitudes stresses in mandibular bone were observed in the cortical area;
- The cancellous bone presented low stress concentration for both geometries;
- The use of implant/bone interface with lower stiffness was capable to diminish or to delay the loads transmitted to the bone.

References

1. KOCA OL, ESKITASCIOGLU G, USUMEZ A. Three dimensional finite element analysis of functional stresses in different bone locations produced by implants placed in the maxillary posterior region of the sinus floor. *J. Prosth. Dent*, 2005, vol. 93, pp. 38-44. DOI: 10.1016/j.prosdent.2004.10.001
2. PAPAVASILIOU G, KAMPOSIORA P, BAYNE S, FELTON D. Three dimensional finite element analysis of stress distribution around single tooth implants as a function of bony support, prosthesis type and loading during function. *J. Prosth. Dent.* 1996, vol. 76, pp. 633-640. DOI: 10.1016/S0022-3913(96)90442-4
3. COCHRAN DL. A comparison of endosseous dental implant surfaces. *J. Periodontol.* 1999, vol. 70, pp. 1523-1539. DOI: 10.1902/jop.1999.70.12.1523
4. HANSSON S. A conical implant-abutment interface at the level of the marginal bone improves the distribution of the stresses in the supporting bone: an axisymmetric finite element analysis. *Clin. Oral. Impl. Res*. 2003, vol. 14, pp. 286-293. DOI: 10.1034/j.1600-0501.2003.140306.x
5. SPIEKERMANN H. Peri-implant pathology. in Implantology, K.H. Rateitschak and H.F. Wolf, Eds. New York: Thieme Medical, 1995, pp. 318.
6. GERAMY A, MORGANO SM. Finite element analysis of three designs of an implant-supported molar crown. *J. Prosthet. Dent.* 2004, vol. 92, pp. 434-440. DOI: 10.1016/j.prosdent.2004.08.011
7. ISHIGAKI S, NAKANO T, YAMADA S, NAKAMURA T et al. Biomechanical stress in bone surrounding an implant under simulated chewing. *Clin. Oral. Implant. Res.* 2002, vol. 14, pp. 97-102.
8. KAYABAŞ O, YÜZBASIOĞLU E, ERZINCANL F. Static, dynamic and fatigue behaviors of dental implant using finite element method. *Adv. Soft. Eng.* 2006, vol. 37, pp. 649-658.

9. GENG JP, TAN KBC, LIU GR. Application of Finite Element Analysis in implant dentistry: a review of literature. *J. Prosthet. Dent.* 2001, vol. 85, pp. 585-598. DOI: 10.1067/mpr.2001.115251
10. NATALI AN, PAVAN PG, RUGGERO AL. Analysis of bone-implant interaction phenomena by using a numérical approach. *Clin. Oral. Implant. Dent.* 2006, vol. 17, pp. 67-74. DOI: 10.1111/j.1600-0501.2005.01162.x
11. COOPER LF, MASUDA T, YLIHEIKKILÄ PK, FELTON DA. Generalizations regarding the process and phenomenon of osseointegration. *Int. J. Oral. Maxillofac. Implants.* 1998, vol. 13, pp. 163-174. WOS: 000073152000002
12. VAN STEENBERGHE D, TEICIO J, NAERT I, NYS M. Damping characteristics of bone-to-implant implant interface. *Clin. Oral. Implants. Res.* 1995, vol. 6, pp. 31-39.
13. LUMBIKANONDA N, SAMMONS R. Bone cell attachment to dental implants of different surface characteristics. *Int. J. Oral. Maxillofac. Implants.* 2001, vol. 16, pp. 627-636, WOS: 000171601900002.
14. TADA S, STEGAROIU R, KITAMURA E, MIYAKAWA O et al. 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.* 2003, vol. 18, pp. 357-368. WOS: 000183428600005
15. KITAMURA E, STEGAROIU R, NOMURA S, MIYAKAWA O. Influence of marginal bone resorption on stress around an implant - a three-dimensional finite element analysis. *J. Oral. Rehabil*. 2005, vol. 32, pp. 279-286. DOI: 10.1111/j.1365-2842.2004.01413.x
16. VAN OOSTERWYCK H, DUYCK J, SLOTEN JV, VAN DER PERRE G et al. The influence of bone mechanical properties and implant fixation upon bone loading around oral implants. *Clin. Oral. Implants. Res.* 1998, vol. 9, pp. 407-418. DOI: 10.1034/j.1600-0501.1996.090606.x
17. ŞTEŢIU AA, ŞTEŢIU M, BURLIBAŞA M, PERIEANU V et al. FEM analysis of masticatory induced stresses over surrounding tissues of dental implant. *Romanian Biotechnological Letters*. 2019, vol. 24 nr. 3, pp. 472-478, DOI: 10.25083/rbl/24.3/472.478
18. BURLIBAŞA M, ŞTEŢIU AA, MARINESCU DG, ŞTEŢIU M et al. Finite element method analysis of the stress induced upon the dental implant by the mastication process. *Romanian Biotechnological Letters*. 2017 vol. 22 nr. 4, pp. 12706-12714, WOS 000408423600004.