Journal of
Applied and Computational Mechanics

Shahid Chamran
University of Ahvaz

Research Paper

Biomechanical Evaluation of Bone Quality Effect on Stresses at Bone-Implant Interface: A Finite Element Study

B. Taharou[1], A. Merdji[1,2], R. Hillstrom[3], A. Benaissa[1], S. Roy[4], N. Della[1],
A. Aid[1], O.M. Mukdadi[5]

[1] Faculty of Science & Technology, University of Mascara, B.P.305 - Mamounia Road, Mascara (29000), Algeria

[2] Laboratory of Mechanics Physical of Material (LMPM), Faculty of Technology, University of Sidi Bel-Abbes, Algeria

[3] Department of Bioengineering, Tandon School of Engineering, New York University, USA

[4] Department of Mechanical Engineering, SRM Institute of Science and Technology, Chennai-603203, India

[5] Department of Mechanical and Aerospace Engineering, West Virginia University, Morgantown, WV 26506, USA

Abstract. The aim of this study is to evaluate the effect of the alveolar bone quality on von Mises stress at the bone-implant interface during occlusal loading. Four (3D) finite element models of fully osteointegrated 3-mm diameter × 11.5-mm length dental implant indifferent alveolar bone with different cortical bone thickness are created, using SolidWorks computer aided design software. The alveolar bone cortical-spongy bone ratio modelled includes I) 90%-10%, II) 60%-40%, III) 40%-60%, and IV) 10%-90%. These models are then exported to ABAQUS software and stress analyses are run under an occlusal load of 70 N acting on the platform face of the dental implant. Results of this study show that the implants are subjected to similar stress distributions in all models; maximum stress values are confined in the outer cervical plate of the cortical bone around the neck. This could explain bone loss and implant de-osseointegration. Peak stresses are lowest in the model with 90% cortical bone (14.2 MPa) and almost doubled in the model with 10% cortical bone (26.6 MPa). The stress values gradually reduce towards the apical area, demonstrating masticatory force transfer from implant to bone. Furthermore, both cortical and spongy bone structures exhibit highest stress values in the model with thinnest cortical layer. The high interfacial stress concentration near the implant-cortical bone junction could lead to bone failure or implant instability induced by fatigue or overload risk. Results of our study could be a first step towards the development of a clinical pre-operative planning tool for dental implantolgy.

Keywords: Bone quality, Dental implant, 3D finite-element analysis, Stress.

1. Introduction

Brânemark's work on osseointegration revolutionized the field of implant dentistry so much so that dental implantation is becoming increasingly synonymous to prosthetic therapeutics and permanent solutions to the problem of missing teeth, there by substituting the lost physiological functions. Rigid prosthetic dental implants do not include periodontal ligaments, which provide a viscoelastic support to the tooth and promote physiological elastic deformation [1]. Dental implantology success is based on the introduction of the osseointegration concept that has revolutionized this discipline by allowing the development of clinically predictable implant systems.

Dental implants can fail as a result of destruction of the support bone and the loss of implant integration following mastication [2, 3]. An understanding in dental biomechanics could help establish certain principles and guidelines for the design of dental implants. The implant-support bone interface structure is of particular interest, given its vulnerability to the risks of deterioration by overload [4, 5]. It is therefore important that the implants transmit forces without causing damage to the support bone.

Bone-implant interfacial stress is commonly analyzed numerically, using the finite element (FE) method [6-11], to determine the stress level anywhere in the different materials with either linear or non-linear mechanical behavior [9,12]. Finite element method has been demonstrated as a powerful tool for studying complex biomedical systems, which may reduce the burden of complicated and limited in vivo and in vitro studies.

The main goal of this study is to elucidate the impact of bone quality and shape on the long-term durability of dental implants by simulating bone-implant interactions, using FE method. Relationship between the applied forces and the interfacial mechanical stresses generated within the bone-implant system was investigated. This numerical contribution could bring valuable insights into the causes of bone loss around dental implants, which could help improve implant designs surgical fixation techniques, as well as for planning of treatment strategy and collaboration among specialists.
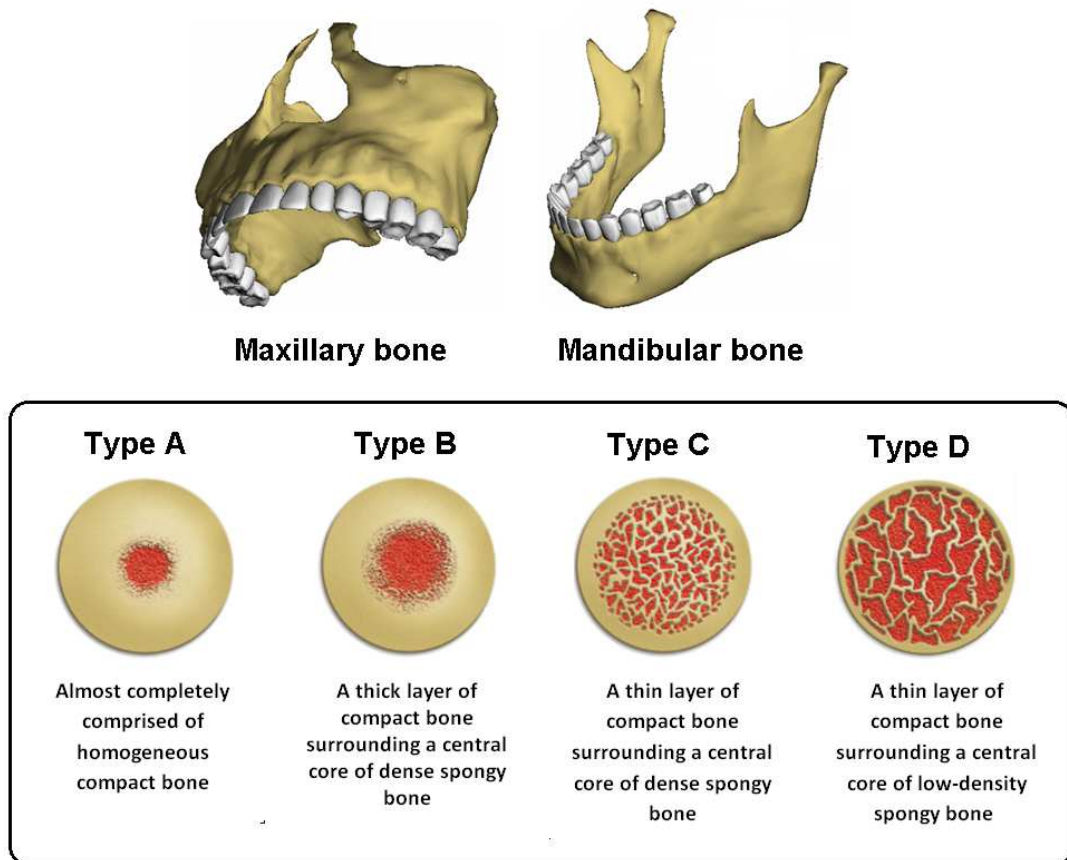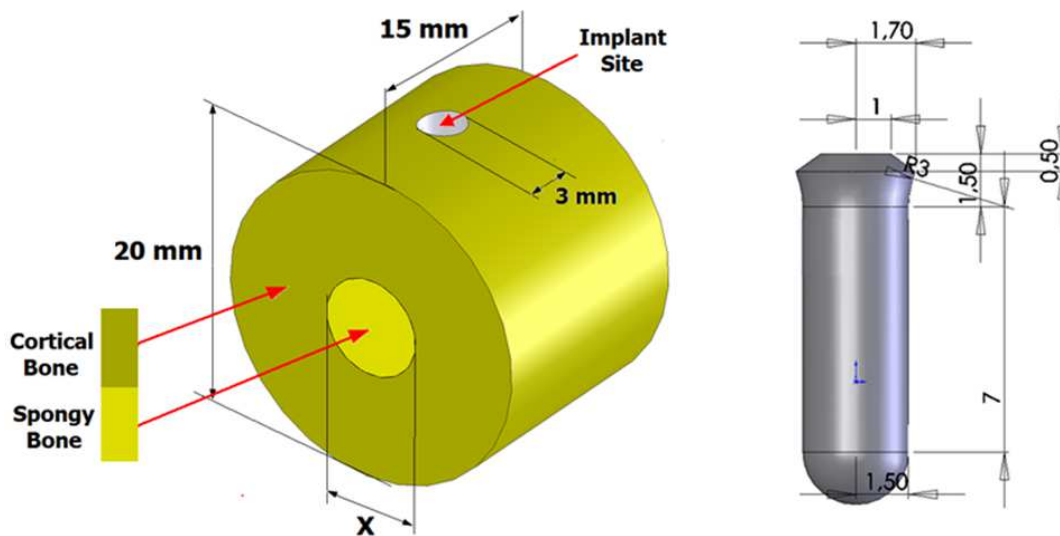
Fig. 1. Bone quality index

Fig. 2. Three-dimensional geometrical model of a dental implant and the surrounding bones

2. Materials and Proposed Methods

Osseointegration durability and stress distribution around the implant is dependent of good quality bone surrounding the implant and a well-maintained interface between the biomaterial of the implant and the bone. Clinically, alveolar bone material properties are categorized into four types as shown in the figure below [14]:

2.1. Geometry

The 3D geometric implant-bone model was created, using SolidWorks software (Dassault Systèmes, Vélizy-Villacoublay, France). The cylindrical implant structure, with diameter = 3 mm, length = 11.5 mm and no thread, was virtually inserted to the bone structure at 8.5 mm from each other. Each system component was modelled individually and assembled to achieve the overall bone-implant system (Fig. 2). The single implant-bone system was simulated by a cylindrical section with diameter = 20 mm diameter and length = 15 mm.

Table 1. Elastic properties of materials used [20-23].

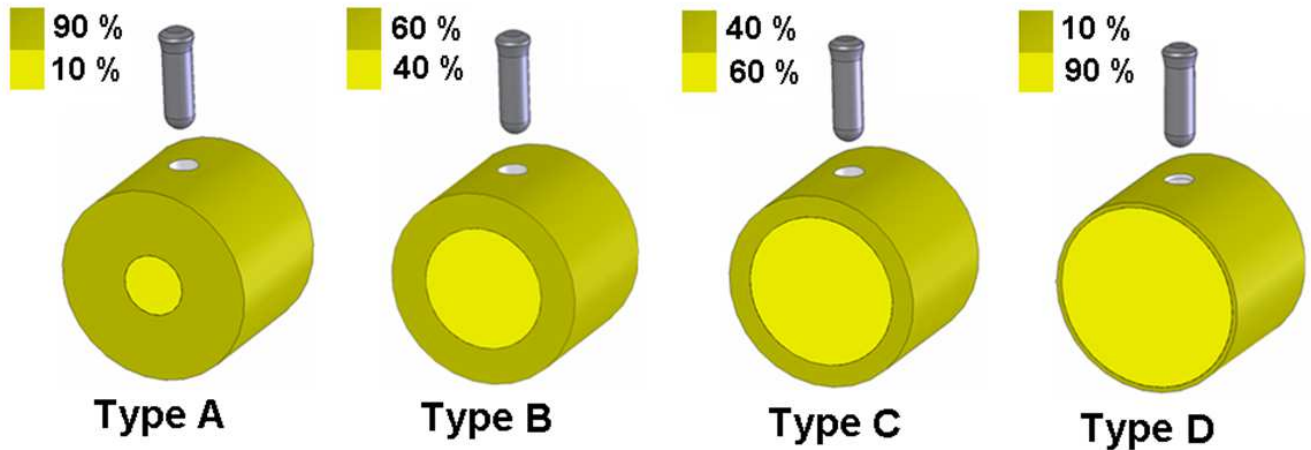| Compounds	Materials	Elastic modulus, E (*GPa*)	Poisson's ratio, *v*
Implant	Titanium	110	0.3
Bone : type A	Cortical bone	13.7	0.3
	Spongy bone	1.37	0.3
Bone : type B	Cortical bone	13.7	0.3
	Spongy bone	1.37	0.3
Bone : type C	Cortical bone	13.7	0.3
	Spongy bone	1.37	0.3
Bone : type D	Cortical bone	13.7	0.3
	Spongy bone	0.270	0.3

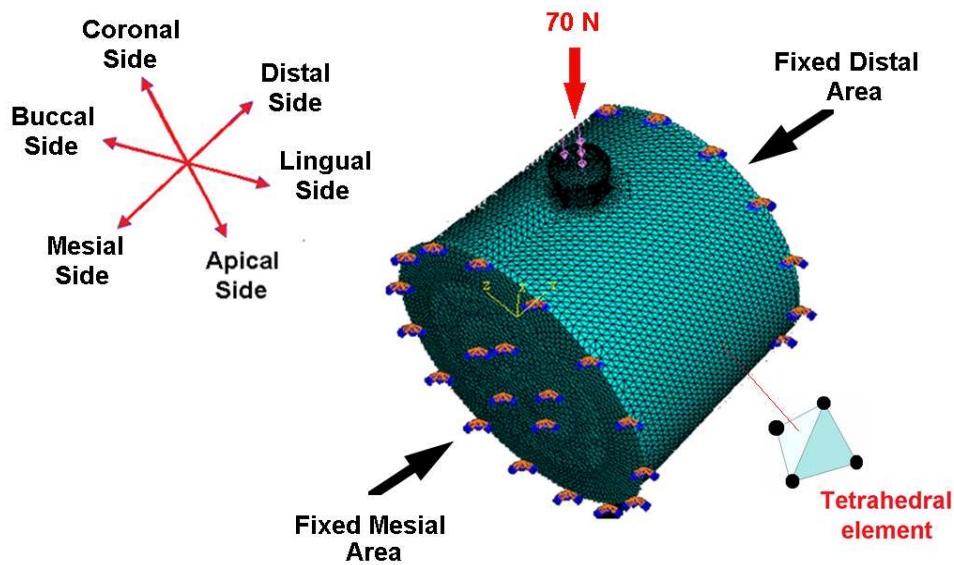**Fig. 3.** Bone qualities considered in this study.

Fig. 4. Finite element model of bone and implant subjected to occlusal static loading

2.1. Bone Quality and Morphology

To investigate the effect of bone quality on stress generated in the alveolar bone, four different bone segment models were proposed, based on its shape and quality, in accordance with published classifications [13-17], as illustrated in Fig. 3:

    I. Type A represents a very compact bone with little spongy bone (90%cortical bone), and the diameter "x" of spongy bone = 6.32 mm.

    II. Type B has a balanced ratio between the spongy layer (40%, dense) and the cortical layer (60%); diameter x = 12.64 mm.

    III. Type C has more spongy bone (60%, dense) than cortical bone (40%). Diameter x = 15.5 mm.

    IV. In type D, the cortical layer is almost absent with 90% of spongy bone of low density. This represents the poorest quality of the alveolar bone, with x = 18.96 mm.

2.3. Material Properties

The parts were exported to FE software ABAQUS v 9.0 (Dassault Systèmes, Vélizy-Villacoublay, France). The required mechanical characteristics of the implant and bone materials (modulus of elasticity and Poisson's ratio) were adopted from previously reported data [20-23] and presented in Table 1. For this comparative study, the implant and bone material properties were assumed to be isotropic and linear elastic.
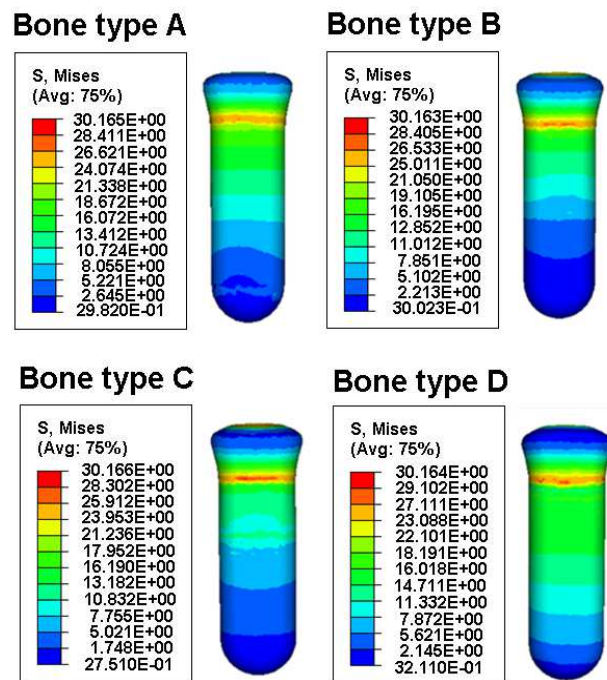
Fig. 5. Von Mises stress distribution on the implant.

2.4. Boundary and Loading Conditions

Complete implant-bone osseointegration was assumed and simulated by merging the nodes on the corresponding surfaces [18, 19]. Identical boundary and loading conditions were used in all the models (Fig. 3). The mesial and distal surfaces of the cylindrical bone segment were mechanically grounded in all directions (Fig. 3). An occlusion load of F=70 N was applied to the top surface of the implant along the vertical axis of implant in the coronal-apical direction [24].

2.5. Meshing

Since the bone-implant interface is subjected to maximum stresses under occlusal loading, a sensitivity analysis was conducted to refine the mesh at this interface until convergence in stress values was reached (Fig. 4). A total of 10,5004 noded tetrahedral elements and 18,590 nodes were used for the FE analyses.

2.6. Analysis

Finite element analyses were run for von Mises stress values in the implant and surrounding bone for all four bone qualities. Stress values for the different bone classifications were compared to evaluate the effect of implant fixation on bone quality.

3. Results and Analyses

Results of von Mises stress generated in the implant and surrounding bones were obtained, based on predefined visual color scale, ranging from dark blue (low stress) to red (high stress), as shown in Figs. 5–7. Equivalent von Mises stress σ_{VM} value is defined in terms of the principal stress state by the following formula [6]:

$$\sigma_{VM} = \frac{1}{2}\sqrt{\left[\left(\sigma_X - \sigma_Y\right)^2 + \left(\sigma_Y - \sigma_Z\right)^2 + \left(\sigma_Z - \sigma_X\right)^2\right]}$$

(1)

where σ_X, σ_Y and σ_Z are principal stresses in X, Y, Z directions.

3.1. Stress Distribution in the Implant

Figure 5 shows that, for all four bone types considered, von Mises stress values were elevated in the implant neck area, particularly at the interface of implant-cortical bone interface. Stress values were gradually reduced towards the apical area of the implant.

3.2. Distribution of stress in the bone

In all four bone conditions, highest stress values were mainly recorded in the outer cervical plate of the cortical bone, adjacent to the implant neck (Fig.6), a critical zone which undergoes masticatory forces via the dental implant. The spongy bone of the alveolar bone in contact with the apex of the implant was subjected to lower stress values compared to the cortical bone (Fig. 7).

Peak stress values were lowest in the cortical bone of type A model than in the other models of types B, C and D. Maximum von Mises stress values for bone type A, B, C and D were 14.230, 19.342, 24.131, and 26.594 MPa, respectively. Compared to type A bone model, we observed increased peak stress values in the cortical bone by 86.9% in type D bone, 69.6% in type C bone, and 35.9% in type B bone models. Moreover, peak stress in the spongy bone of model D, was highest compared to those of models A, B, and C. These stress values decreased progressively away from the interface implant-bone.

Figs. 8, 9 show intersection lines along the bone-implant interface at the cervical, linguo-buccal and disto-mesial sections, respectively. Stress values were plotted at these interfaces to compare variations of stress values in the bone. Graphical representations were used to compare von Mises stress levels under vertical occlusal loading for the four bone-implant systems. The maximum stress values are shown at thresholds P_1, P_2, P_3 and P_4 of the lingual, distal, buccal and mesial sides, respectively. In this study, all bone-implant models have an orthogonal symmetry to the implant symmetrical axis, for this reason only the result plots of dental half-planes, buccal-lingual and distal-mesial are presented. The implant is not shown clarity purposes.
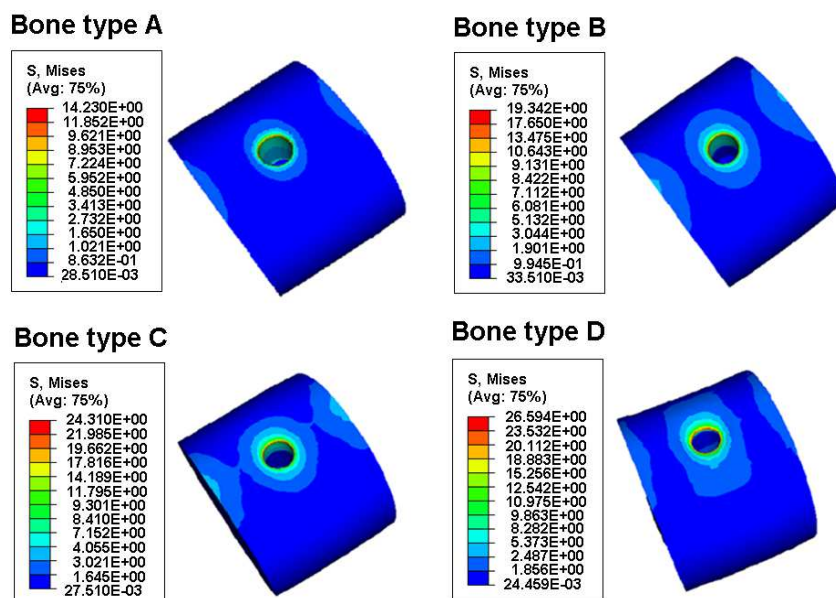
Fig. 6. Von Mises stress distribution on the cortical bone
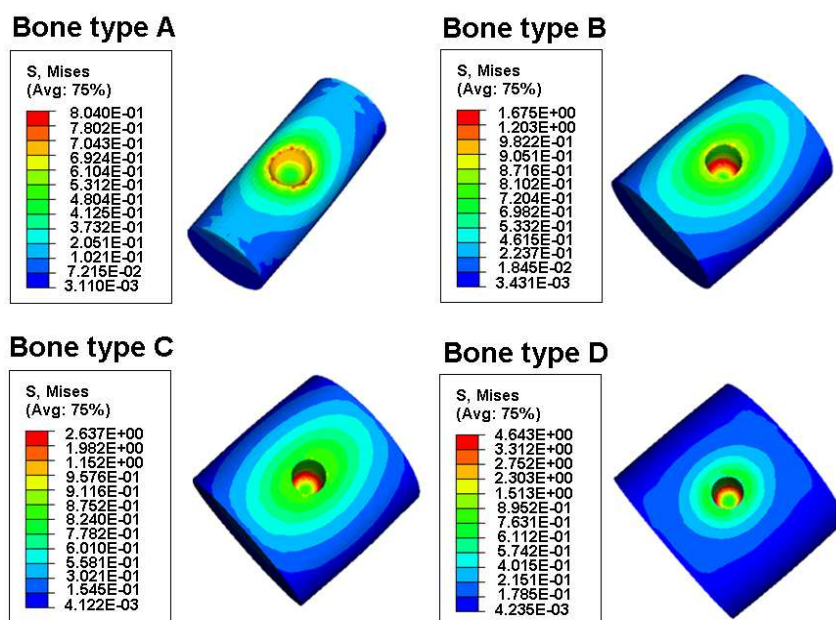

Fig. 7. Von Mises stress distribution on the spongy bone.

Finite element analysis results show uneven stress distribution along the cervical external line of the bony socket-loaded implant interface for all four bone type models. Figure 6 shows highest stress concentrations at points P2 and P4 of both distal and mesial sides, respectively. Figure 8 shows highest stress values for type D bone compared to the other three bone types.

Fig. 9.a shows the variations of equivalent von Mises stresses along the cortical bone–implant interface and is presented with respect to the disto-mesial and linguo-buccal interfacial paths, respectively, for all four bone types. The stress distributions show a decrease in stress level in the corono-apical direction and an increase at the cortical bone-spongy bone junction in the points $D_{1/Cortical}$, $C_{1/Cortical}$, $B_{1/Cortical}$, and $A_{1/Cortical}$ of the disto-mesial line. Similar behaviors are observed at the points $D_{3/Cortical}$, $C_{3/Cortical}$, $B_{3/Cortical}$ and $A_{3/Cortical}$ of the linguo-buccal line.

Fig. 9.b shows the von Mises stress distributions at the implant–spongy bone interface line in the disto-mesial and linguo-buccal directions, respectively. For both directions, stress values decrease towards the apical side, then suddenly increased at the implant apex-spongy bone interface. Stress levels in the spongy bone increases as cortical bone thickness decreases. The spongy area of type D bone is subjected to highest stress levels during occlusal loading, compared to bone types A, B and C. Results of the computational models show that stress values in bones of types A and B are generally lower than those in types C and D.

4. Discussion

A method for evaluating the effect of the alveolar bone quality on von Mises stress in the implant and bone was presented. Von Mises stress was computed as it is commonly used in biomechanical studies [23, 25]. Ductile material failures are generally predicted using the von Mises criterion, which is based on the distortion energy of the material. All principle stresses are involved in the calculation of von Mises stress, thereby allowing a better representation of stress generated.
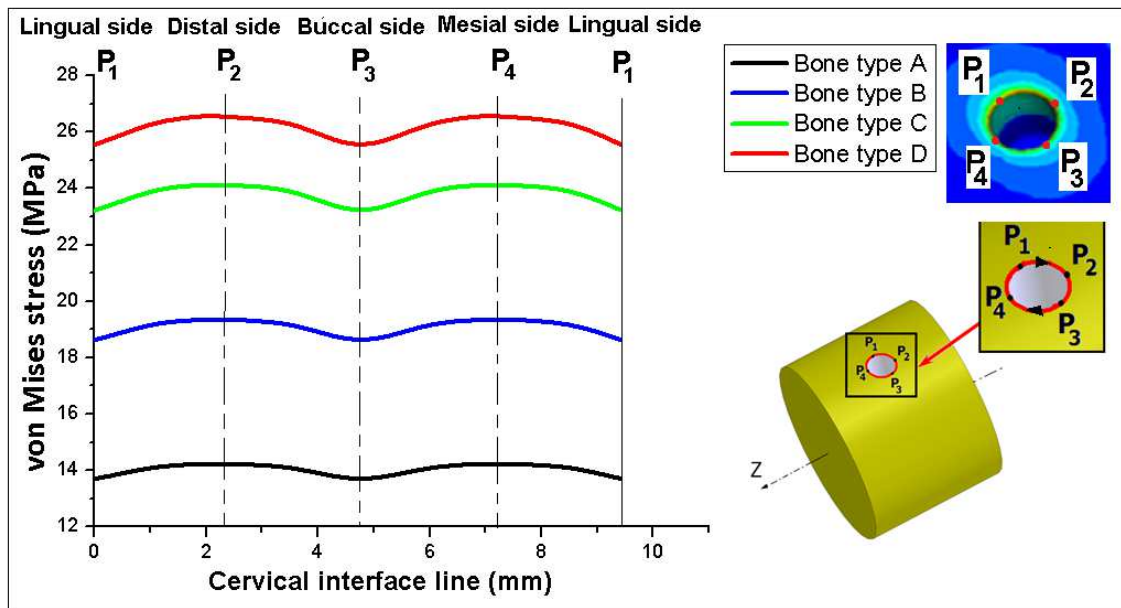
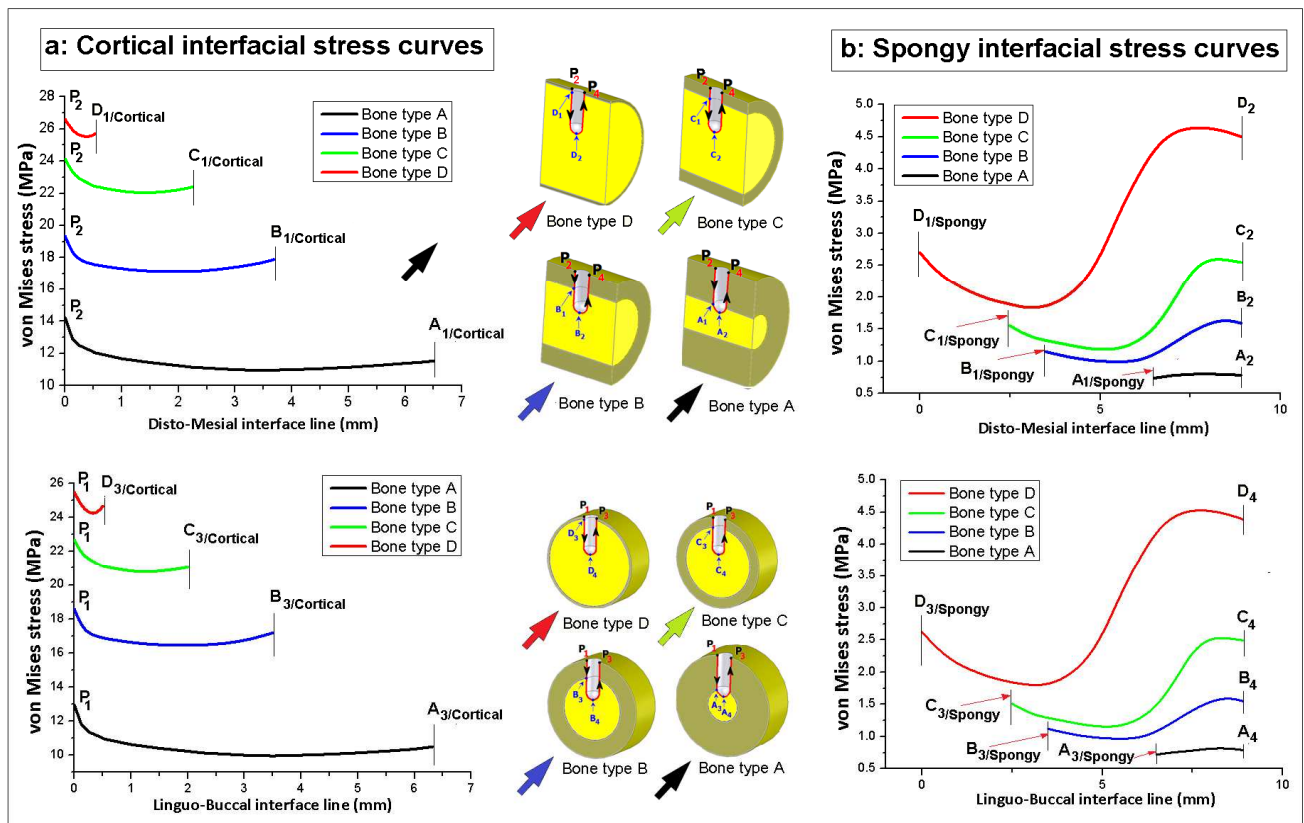**Fig. 8.** Von Mises stress values along the cervical line at the implant-bone interface.

Fig. 9. Von Mises stress values at linguo-buccal and disto-mesial interface lines:
(a) Cortical interfacial stress curves, (b) Spongy interfacial stress curves.

High stress values in the neck area of the implant-cortical bone interface, which gradually reduced towards the apical area, demonstrates how masticatory forces are transferred from the implant to the bone [24-26]. The implant is subjected to higher stress values, compared to the mandibular bone because its modulus of elasticity (E = 110000 MPa is about 8 times and 80 times higher than those of cortical bone (E = 13700 MPa) and spongy bone (E = 1370 MPa), respectively. These findings demonstrate that the occlusal loads acting on the implant are transferred directly to the surrounding bone [25, 28].

The high interfacial stress concentration near the implant-cortical bone junction, could lead to bone failure or implant instability induced by fatigue or overload risk. Indeed, several researchers indicated that occlusal overload is critical and often caused bone loss and de-osseointegration of implants [25, 27]. The cortical bone around dental implants provide a support zone for occlusal forces, suggesting that implants could be more vulnerable to bone loss by elevated mechanical stresses, in particular for bones of low mechanical properties such as in bone types C and D. Because of the big difference between the mechanical properties of both cortical and spongy bone, von Mises stresses in these regions are shown distinctly for better comparison [25, 27].
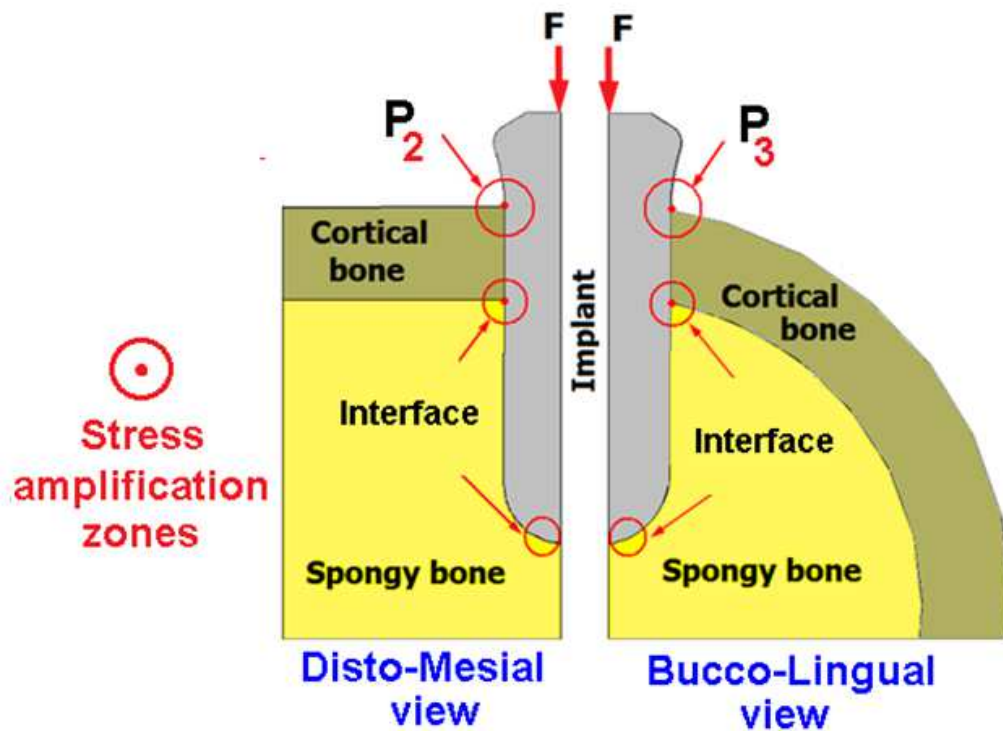
Fig. 10. Stress amplification zones.

Cortical bone is approximately ten times more rigid than spongy bone and is thicker in model A compared to the other models. This allows better load transfer from the implant-cortical bone interface compared to models with thinner cortical bones that result in more concentrated stress areas. As a result, the spongy bone in model A is subjected to lower peak stress compared to the other models with thinner cortical bones.

Bone modeling and remodeling is a complex physiological process that is controled by the level of osteoblast and osteoclast cell activities, which are sensitive to variation in mechanical stress levels. Therefore, bone loss mechanistic occurs due to an imbalance between the amount of bone resorbed by osteoblast cells and the amount of bone formed by osteoclast cells [28]. This explains the bone loss at the implant-cortical bone interface, where stress concentrations are high. Therefore, attention should be given to the implant contact area with the cortical layer of the bone [25-27]. Several radiology studies reported that the neck area around the implant experience significant bone loss [25, 30]. This is in agreement with our results obtained from the finite-element method, confirming that the cortical bone area adjacent to the implant exhibit the highest stress levels in the bone-implant system.

From the medical point of view, repetitive occlusal loadings are likely to cause micro damage accumulation in the bone adjacent to the implant [30, 31]. The phenomenon of crestal bone loss by occlusal loading effect may result in a favorable environment for anaerobic bacterial growth that may lead to serious progressive peri-implant bone resorption and, eventually, more bone destruction [18, 19]. This insufficient bone support is critical, may deter implant stability, and increase the risk of implant de-osteointegration.

The failure of a dental implantation due to physiological masticatory loading on the implant may occur several years after successful osteointegration and installation. It is characterized by a loss of attachment and de-osseointegration. It can be either progressive or declares itself rapidly. De-osseointegration of the implant may result in significant peri-implant mobility, pain and/or bone loss. This phenomenon has been observed in bone types C and D [18, 26].

Stress values at the implant-bone interface cannot be measured in vitro, but can be predicted by computational methods. Therefore, computational models of the four different bone types were developed to compare the influence of masticatory loads on interfacial stress distribution in the bone and help understand the biomechanical behavior of different bone qualities following dental implantology. The constraints used in the models took into consideration the effect of an axial occlusal load in all directions because the mechanical behavior of the bone in different directions is different [22].

Stress concentrations at points P_2 and P_4 along the cervical line of the bone socket-implant interface were also observed in other studies, even for occlusal combined loads [32]. However, this phenomenon of stress concentrations in distal and mesial sides is not well understood. Based on knowledge of our previous studies [1, 25, 33], this phenomenon may be justified, a priori, by the applied boundary conditions to the implant-bone system, as well as by the cylindrical shape of the bone.

Hence, the bone axis (P_2 P_4) can be considered as the generator of a cylinder. When a vertical load is applied, the action of this load is total, unlike along the axis (P_1 P_3) which is tangential to the cylinder, so the action of the vertical force applied to the implant is manifested by a tangential component only. For this reason, both points P2 and P4 present the highest stress values in the bone which return to plasticity due to the risk of overloading risk or excessive masticatory load. This can be observed clinically by bone loss in areas P_2 and P_4. The highest stress level predicted for Type D bone suggests that dental implants are most likely to fail when the cortical bone is thin.

The high stress levels at the cortical-spongy bone junctions in Figs 8 and 9 are attributed to the geometry at the interface, as indicated in Fig. 10. On the other hand, when two materials, with differentelastic moduli, are bonded with no intervening material and one of them is loaded, an increased stress level is observed at the interface. Therefore, during masticatory loading, the cortical bone becomes an area of interaction between two biomechanical stress fields - the external cervical stress and the internal inter-bone stress. When the two stress fields are very close to each other and when the cortical layer thickness is decreased, this stress interaction in the cortical layer is amplified, which might lead to implant loosening. From the clinical point of view, these mechanical interactions are reflected by specific signs and symptoms of significant vertical bone loss associated with the formation of a peri-implant pocket.

Results of the four bone models show that that stress values in the spongy part are very low compared to those in the cortical part. This is because there is higher load transfer in materials with high elastic modulus. Hence, bone loss is localized at the cortical bone-implant neck interface, without any involvement of the spongy part around the implant in apical side which remains osteointegrated. So, the destruction of spongy bone around the implant can become progressive by occlusal load effects as a function of time. The ability to reduce the bone loss can result in major esthetic and clinical benefits. Hence, further studies to maintain the cortical bone is warranted. The optimization of implant neck geometry, which is surrounded by the cortical bone layer, is a technique that requires additional retrospective and long-term studies in the dental implantolgy field. Further clinical studies are required to better understand the risks that may lead to the progression of bone loss and to develop more effective techniques for antibacterial therapy and regenerative surgical treatment.

The lower stress values generated in bone types A and B during simulated masticatory loading indicate lower risk of lesions in thick cortical bones. These results agree qualitatively with previously reported studies [25-34], although the implant geometries and boundary conditions used in the other finite-element models are different from those adopted in our study. It remains to be emphasized that the choice of implant geometry is not only due to the need to reduce stress levels, but is sometimes limited by anatomical complications [18, 19].

Analyses with a variation of the bone quality have also been achieved [15-17]. They have proved that these parameters have a strong influence on the way of interfacial stress distributions in the bone and the implant stability. Indeed the junction cortical bone-spongy bone plays a significant role in the mechanical behavior of the studied system, leading to a stress amplifications of implants which can be appreciated through the stress field and amplitude.

In this study, the ideal condition of full osseointegration between implant and bone is considered. Stress analyses were carried out, assuming a confined static load, and bone tissues were modeled as liner isotropic homogenous elastic materials. These do not constitute real clinical conditions due to possible defects in osseointegration between bone and implant, time dependencies, functionally distributed loads, and material anisotropy, non-homogeneity as well as the non-linear behavior of the bone [17, 19, 25]. Nevertheless, these limitations do not encumber the relevance of results of this study, which agree well with a number of well-established studies [28, 30]. The assumptions used in this study can be considered acceptable since they provide useful and clinically acceptable indications.

The finite element method offers an efficient procedure for analyzing complex biomedical systems, such as the bone-implant structure adopted in this study, under a variety of alveolar bone anatomy, occlusal loading, excitation, and boundary conditions [25-28]. Results of this study show that finite element method is a powerful mathematical technique for in vivo and in vitro simulations. Numerical results obtained from finite element method could help prosthodontists understand the physiological consequences and relative performances of treatment protocols. Such models could be used as a pre-operative planning guide to predict complex biomechanical interactions developed at the implant–bone and cortical-spongy bone interfaces. This could help prosthodontists make informed decisions on surgical treatment techniques for different bone qualities.

Depending on this study results, different issues have been investigated with finite element method ranging from more clinically oriented topics related to the bone quality (e.g., knowing the reasons that lead to bone loss and implant risk) to more fundamental problems dealing with the mechanical aspects of biological processes (e.g., stress and strain around osteocyte lacunae) as well as with the biomechanical behavior of bone at its ultrastructure. A better understanding of the relationship between bone structure and mastication loading is expected to be important for the current trends in implant interfaces design for each type of bone, where the structure of bone is considered as a possible source of inspiration, as well as for more successful approaches to know the peri-implant environment and its relevance to long-term success of osseointegrated implants.

On the basis of the results obtained, the search for reasonable solutions making it possible to reduce these implant interfacial constraints has become a very important research axis. Thus, from a biomechanical point of view, a key factor for the predictability of implant protocols is the development of implants and prosthesis designs capable of providing some degree of stability, under masticatory standard loading. Several alternatives have been studied, including in particular variations in implant positioning, implant design, prosthesis geometry, occlusive loading conditions, prosthetic components and biomaterials used[16, 18-20].

For several years, numerous studies have been carried out to obtain an implant surface which induces osteointegration of better quality and faster than the smooth surface [28, 31]. The emergence of implants with a rough and porous surface at the same time, aims to improve the quality of osseointegration by increasing the implant surface. This optimizes the extent of the bone-implant interface surface with more contact, which will result in faster, more stable bone anchoring. This quantitative and qualitative gain in bone-implant contact is indeed an undeniable asset in sparse bones such as type C and D bones.

5. Conclusions

Results of the finite element analyses showed that the alveolar bone quality is a major parameter that affects interfacial stress distributions around endosseous implants. Within the limitations of this study, the following conclusions may be drawn:

1. The long-term success and stability of a dental implantology is dependent upon continuous interfacial osseointegration and preservation of bone quality around the implant.
2. Risk factors including elevated mechanical stresses generated from masticatory forces, increased with rigidity of the implant-bone system.
3. Jawbone anatomies with thin cortical bone can pose significant risks to patients, including bone loss and de-osseointegration of implants.

On the other hand, our findings revealed that thinner cortical bones were subjected to higher stress values, which agree with the findings of previous researchers. Moreover, this study provides a clear biomechanical explanation of this phenomenon, especially the high stress levels developed as a result of stress interaction fields between cortical-spongy bone junction and cortical bone-implant neck interface along the external cervical line, and their relationships to the alveolar bone quality. This phenomenon was a major cause of cortical bone loss around the implant neck.

6. Future Outlook

No matter previous dental implant research in vitro or in vivo, all the interfaces are normal bone tissues. In clinical reality, the patients may have dysfunction of immune system or other diseases. Whether the loading effect on the abnormal conditions is similar to previous research remains unknown. We still need to explore in our future research. Besides, the effects of interfacial stresses on cells are still not clear for the limitation of cell cycle and the real mastication frequency and time. Therefore, we still need to establish the new mode for cell research under fretting condition. The mechanism of mechanical stress produced by mastication loads on implant/bone interface still needs to be further elaborated.

Author Contributions

Conflict of Interest

Funding

References

[1] Merdji, A., Mootanah, R., Bachir Bouiadjra, B., Benaissa, A., Aminallah, L., OuldChikh, B., Mukdadi, S., Stress analysis in single molar tooth, *Materials Science and Engineering C*, 33(2), 2013, 691-698.

[2] Lang, L.A., Kang, B., Wang, R.F., Lang, B.R., Finite element analysis to determine implant preload, *Journal of Prosthetic Dentistry*, 90(6), 2003, 539-546.

[3] Waters N.E., Some mechanical and physical properties of teeth. *Symposia of the Society for Experimental Biology*, 34, 1980, 99-135

[4] Kim, S.J., Kim, S., Choi, H., Woo, D., Park, Y.B., Shim, J.S., Kim Shim, J.S., Kim, H.S., Lee, K.W., A three dimensional finite element analysis of short dental implants in the posterior maxilla, *The International Journal of Oral & Maxillofacial Implants*, 29(2), 2014, 155-164.

[5] Kozlovsky, A., Tal, H., Laufer, B.Z., Leshem, R., Rohrer, M.D., Weinreb, M., Artzi, Z., Impact of implant overloading on the peri-implant bone in inflamed and non-inflamed peri-implant mucosa, *Clinical Oral Implants Research*, 18(5), 2007, 601–610.

[6] Tang, C.B., Liu, S.Y., Zhou, G.X., Yu, J.H., Zhang, G.D., Bao, Y.D., Wang, Q.J., Nonlinear finite element analysis of three implant–abutment interface designs, *International Journal of Oral Science*, 4(2), 2012, 101–108.

[7] Merdji, A., Della, N., Benaissa, A., Bachir Bouiadjra, B., Serier, B., Mootanah, R., Muslih, I., Mukdadi, O.M., Numerical analysis of dental caries effect on the biomechanical behavior of the periodontal system, *Journal of Nanotechnology in Engineering and Medicine*, 6(3), 2015, 031004.

[8] Chen, X., Mao, B., Zhu, Z., Yu, J., Lu, Y., Zhang, Q., Yue, L., Yu, H., A three-dimensional finite element analysis of mechanical function for 4 removable partial denture designs with 3 framework materials: CoCr, Ti-6Al-4V alloy and PEEK, *Scientific Reports*, 9 (1), 2019, 13975.

[9] Wu, T., Liao, W., Dai, N., Tang. C., Design of a custom angled abutment for dental implants using computer-aided design and nonlinear finite element analysis, *Journal of Biomechanics*, 43(10), 2010, 1941–1946.

[10] Kitagawa, T., Tanimoto, Y., Nishiyama, N., Aida, M., Application of finite element analysis for taper implant-abutment joints in dental implant systems, *International Journal of Oral-Medical Sciences*, 7(1), 2008, 1-6.

[11] Javed, F., Romanos, G.E., The role of primary stability for successful immediate loading of dental implants: A literature review, *Journal of Dentistry*, 38(8), 2010, 612–620.

[12] Holmgren, E.P., Seckinger, R.J., Kilgren, L.M., Mante, F., Evaluating parameters of osseointegrated dental implants using finite element analysis–a two-dimensional comparative study examining the effects of implant diameter, implant shape, and load direction, *Journal of Oral Implantology*, 24(2),1998, 80–88.

[13] Siegele, D., Soltresz, U., Numerical investigations of the influence of implant shape on stress distribution in the jaw bone, *International Journal of Oral & Maxillofacial Implants*, 4(4),1989, 100–113.

[14] Branemark, P.I., Zarb, G.A., Albrektsson, T., *Tissue-integrated prostheses: Osseointegration in clinical dentistry*, Quintessence Publishing Company, Chicago, USA, 1985.

[15] Holmes, D.C., Loftus, J.T., Influence of bone quality on stress distribution for endosseous implants. *Journal of Oral Implantology*, 23(3), 1997, 104–111.

[16] 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. International, *Journal of Oral & Maxillofacial Implants*, 18(3), 2003, 357–368.

[17] Lee, S., Gantes, B., Riggs, M., Crigger, M., Bone density assessments of dental implant sites: 3. bone quality evaluation during osteotomy and implant placement, *International Journal of Oral & Maxillofacial Implants*, 22(2), 2007, 208–212.

[18] Lee, C.C., Lin, S.C., Kang, M.J., Wu, S.W., Fu, P.Y., Effects of implant threads on the contact area and stress distribution of marginal bone, *Journal of Dental Sciences*, 5(3), 2010, 156-165.

[19] Rasouli, G.A.A., Geramy, A., Yaghobee, S., Khorsand, A., Yousefifakhr, H., Rokn, A., Soolari, A., Evaluation of platform switching on crestal bone stress in tapered and cylindirical implants: afinite element analyses. *Journal of the International Academy of Periodontology*, 17(1), 2015, 2-13.

[20] Bozkaya, D., Muftu, S., Muftu, A., Evaluation of load transfer characteristics of five different implants in compact bone at different load levels by finite elements analysis. *Journal of Prosthetic Dentistry*, 92(6), 2004, 523–530.

[21] Chun, H.J., Park, D.N., Han, C.H., Heo, S.J., Heo, M.S., Koak, J.Y., Stress distributions in maxillary bone surrounding overdenture implants with different overdenture attachments. *Journal of Oral Rehabilitation*, 32(3), 2005, 193–205.

[22] Ishigaki, S., Nakano, T., Yamada, S., Nakamura, T., Takashima, F., Biomechanical stress in bone surrounding an implant under simulated chewing, *Clinical Oral Implants Research*, 2003, 14(1), 97-102.

[23] Van Staden, R.C., Guan, H., Loo, Y.C., Application of the finite element method in dental implant research, *Computer Methods in Biomechanics and Biomedical Engineering*, 9(4), 2006, 257–270.

[24] Al-Sukhun, J., Kelleway, J., Helenius, M., Development of a three-dimensional finite element model of a human mandible containing endosseous dental implants, I. Mathematical validation and experimental verification, *Journal of Biomedical Materials Research Part A*, 80(1), 2007, 234-46.

[25] Merdji, A., Bachir Bouiadjra, B., Ould Chikh, B., Mootanah, R., Aminallah, L., Serier, B., Muslih I.M., Stress distribution in dental prosthesis under an occlusal combined dynamic loading, *Materials & Design*, 36, 2012, 705-713.

[26] Akca, K., Cehreli, M.C., Biomechanical consequences of progressive marginal bone loss around oral implants: a finite element stress analysis, *Medical & Biological Engineering & Computing*, 44(7), 2006, 527-535.

[27] Correia, A.R., Piloto, P., Campos, J.C., Vaz, M., Finite element analysis of the mechanical behavior of a partially edentulous mandible as a function of spongy bone density, *Revista Odonto Ciência*, 24(1), 2009, 22-27.

[28] Santiago, J.F., Verri, F.R., de Faria Almeida, D.A., de Souza Batista,V.E., Lemos, C.A., Pellizzer, E.P., Finite element analysis on influence of implant surface treatments, connection and bone types, *Materials Science and Engineering: C*, 63(1), 2016, 292–300.

[29] Beikler, T., Flemmig, T.F., Implants in the medically compromised patient, *Critical Reviews in Oral Biology & Medicine*, 14(4), 2003, 305-316.

[30] Lofaj, F., Kucera, J., Nemeth, D., Kvetkova, L., Finite element analysis of stress distributions in mono- and bi-cortical dental implants, *Materials Science and Engineering: C*, 2015, 50:85–96

[31] Enwei, Z., Fei, G., Analysis of static force and fatigue between thread structure of dental implant and contact surface. *Journal of Clinical Rehabilitative Tissue Engineering Research*, 14(30), 2010, 5531–5534.

[32] Goiato, M.C., Pellizzer, E.P., da Silva, E.V., Bonatto, L., dos Santos, D.M., Is the internal connection more efficient than external connection in mechanical, biological, and esthetical point of views?A systematic review, *Oral and Maxillofacial Surgery*, 19(3), 2015, 229-242.

[33] Ammar, H.H., Ngan, P., Crout, R.J., Mucino, V.H,, Mukdadi, O.M., Three-dimensional modeling and finite-element analysis in treatment planning for orthodontic tooth movement, *American Journal of Orthodontics and Dentofacial Orthopedics* , 139(1), 2011, 59-71.

[34] Von Recum, A., *Handbook of Biomaterials Evaluation: Scientific, Technical and Clinical Testing of Implant Materials*, MacMillian, New York, 1986.

ORCID iD

Belaid Taharou https://orcid.org/0000-0001-7014-5027
Ali Merdji https://orcid.org/0000-0003-1280-9723
Rajshree Hillstrom https://orcid.org/0000-0002-2365-5486
Ali Benaissa https://orcid.org/0000-0002-7566-7171
Sandipan Roy https://orcid.org/0000-0002-6888-272X
Noureddine Della https://orcid.org/0000-0002-6759-6916
Abdelkrim Aid https://orcid.org/0000-0001-6355-958X
Osama Mukdadi https://orcid.org/0000-0001-9250-0720

Publisher's Note Shahid Chamran University of Ahvaz remains neutral with regard to jurisdictional claims in published maps and institutional affiliations.