ISBN 978-93-84468-19-4
Proceedings of 2015 International Conference on Bio-Medical Engineering and Environmental Technology (BMEET-2015)
London, March 21-22, 2015, pp. 186-192

The Interest of Damping Materials in the Stability of Dental Implants in Alveolar Bone

Ali Merdji[1,2], Rajshree Mootanah[2], Laid Aminallah[1], Bel-Abbes Bachir Bouiadjra[1], Boualem Serier[1], and Osama Mukdadi[3]

[1]Laboratory of Mechanical and Physical of Materials (LMPM), Djillali Liabes University, Sidi Bel-Abbes, Algeria

[2]Medical Engineering Research Group, Faculty of Science & Technology, Anglia Ruskin University Bishop Hall Lane, Chelmsford, Essex, UK

[3]Department of Mechanical and Aerospace Engineering, West Virginia University, USA

Abstract**:** *It has been well known that the success of dental implant is heavily dependent on initial stability and long-term Osseo-integration due to optimal stress distribution in the surrounding bones. For this reason, the search of the rational solutions to reduce these stresses has become an important issue in this field. This study describes a numerical study performed with the finite element method of new dental implant system. A conventional Brånemark dental implant system was redesigned and a damping material (elastomer) was interposed between the abutment and the framework crown. The goal was to attenuate the loading of the bone surrounding the implant with high magnitude stresses. The new design was assessed and the von Mises interface stresses compared with the ones provoked by the conventional implant. Generally, the new implant provoked lower interface stresses due to the stress shielding effect of the damping material.*

Keywords: *Dental implant, damping material,, finite element method, stress*

1. Introduction

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 [2] 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 [3]. This overloading would cause the microdamage accumulation at bone and results in primary marginal bone loss [4-6].

Finite element analysis (FEA) has been used extensively to predict the biomechanical performance of various dental implant and prosthesis designs as well as the effect of clinical factors on implant success. By understanding the basic theory, method, application, and limitations of FEA in implant dentistry, the clinician will be better equipped to interpret the results of FEA studies and extrapolate these results to clinical situations [5]. FEA have been used to study the effects of various shapes of prostheses and dental implants on distribution of stresses generated in the surrounding jaw bone and to determine an optimal thread shape for better stress distribution. [11-14]

The purpose of this study was to evaluate the effects of adding a bio-elastomer to the prosthetic components of implant system on stress distribution under vertical and horizontal loading conditions. The bio-elastomer was interposed between the abutment and the framework crown in order to damp the occlusive shocks and to attenuate the stress concentrated at the implant/bone interface.

http://dx.doi.org/10.17758/UR.U0315239

2. Materials and methods

2.1. Geometrical Models

A posterior mandible segment with an implant and a superstructure were modelled on a personal computer, using a CAD software. A cross-section of a mandible in the first molar region was used as a basis for a solid model, and then the cross-sectional image was extruded to create a three-dimensional mandible segment. This section was modelled 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.1). In this study two different types of implant system were compared:

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

The elastomer represented a considerably small component as 0.5 mm thickness of the implant system and was assimilated into the volume of the framework.

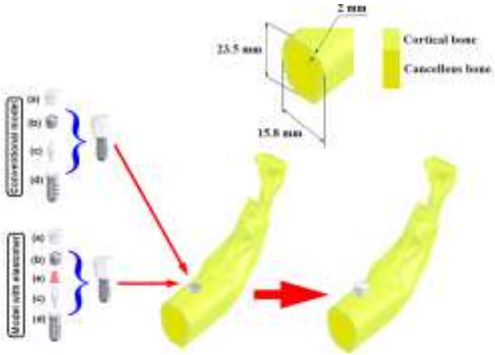


Fig. 1: Components of the models

2.2. Material properties

All materials used in the models were considered to be isotropic, homogeneous, and linearly elastic. The elastic properties were taken from the literature, as shown in Table 1

TABLE I: Material properties used in analysis [10-16]

Parts	Materials	Elastic modulus, E (GPa)	Poisson's ratio
Crown	Feldspathic porcelain	82.8	0.33
Framework	Co–Cr alloy	218	0.33
Elastomer	Silicone	0.006	0.49
Abutment	Titanium	110	0.3
Implant	Titanium	110	0.3
Mandibular bone	Cortical bone	14.5	0.323
	Cancellous bone	1.37	0.3

2.3. Boundary conditions and finite element model

In order to define the boundary conditions, a 3D coordinate system was defined by three loads in the coronal-apical direction, lingual-buccal direction and mesial-distal direction. For the boundary conditions, 3 zones were considered (Fig. 2.a):

- The inferior plane of the mandibular bone was defined as having zero displacement.
- The most coronal plane of the crown was subjected to a load of 3 MPa in either the lingual-buccal or mesial-distal directions or a load of 10 MPa in coronal-apical direction.

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

The implant was rigidly anchored in the bone model along its entire interface. The same type of contact was provided at the prosthesis-abutment interface.

The mesh of the components is simplistic and consisted of linear tetrahedron elements with four nodes (Fig. 2.b). 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.

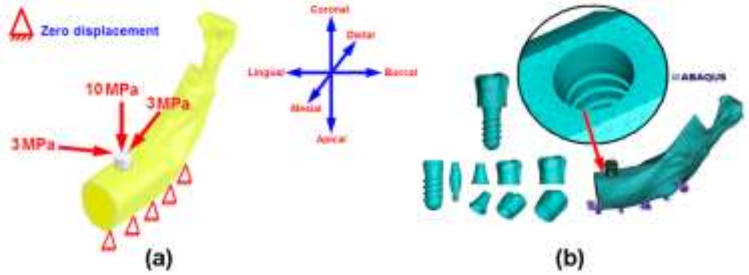


Fig. 2: Boundary conditions (a) and mesh model (b)

3. Result

The distributions of overall stress state for each component in both models 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. The Maximum stress values in each component are shown in Fig.9.

3.1. Implant

Fig. 3 represents the von Mises stresses distribution within the implants. In the conventional model, maximum von Mises stresses were concentrated in the implant at the neck region; except the implant of second model with elastomer under both horizontal loadings, the stresses were concentrated at the abutment implant connection area.

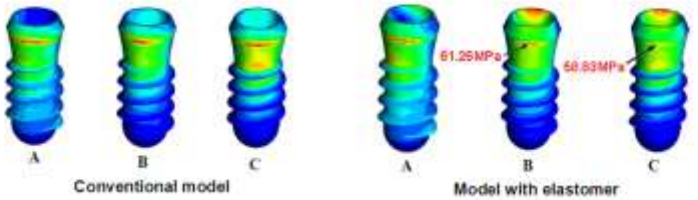


Fig. 3: Distribution of stresses on the implant in both models.
A. Coronal-apical load, B. Lingual-buccal load, C. Distal-mesial load

3.2. Framework

Fig.4 represents the von Mises stresses distribution within the frameworks. For coronal-apical loading, stresses were located at the framework implant connection area on the buccal cervical margin. For horizontal loading, stresses were located at the same area but on the buccal lingual cervical margin or distal mesial cervical margin.

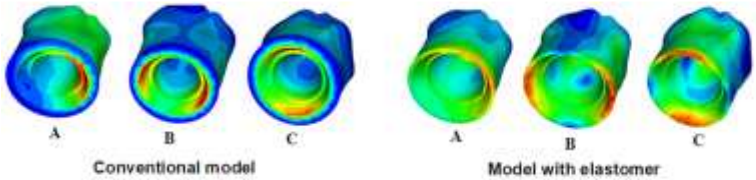


Fig .4: Distribution of stresses on the framework.in both models
A. Coronal-apical load, B. Lingual-buccal load, C. Distal-mesial load.

http://dx.doi.org/10.17758/UR.U0315239

3.3. Occlusive surface

The maximum von Mises stresses on the occlusal surface were concentrated at the top surface of crown on the central furrow for coronal-apical loading. The maximum stresses were located on distal and mesial furrow for lingual-buccal loading, and on lingual and buccal furrow for distal-mesial loading (Fig. 5).

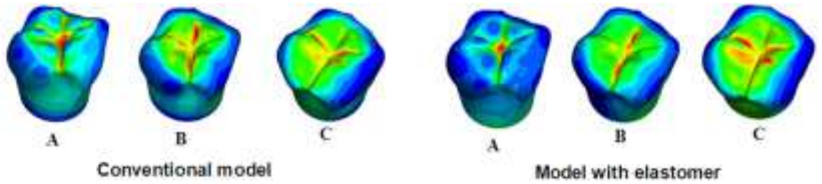


Fig. 5: Distribution of stresses on the occlusal surface material.in both models
A. Coronal-apical load, B. Lingual-buccal load, C. Distal-mesial load

3.4. Abutment

Fig.6 represents von Mises stress distribution within the abutment. In the conventional model, the maximum von Mises stresses on the abutment were concentrated at the implant abutment area. In the second model with elastomer, maximum von Mises stress was concentrated in the abutment, near to the abutment implant junction. For coronal-apical loading the maximum stress was generated at the buccal side.The maximum amount of Von Mises stress was concentrated on buccal and lingual regions of abutment due to loading in lingual - buccal direction, and on mesial and distal sides of abutment due to loading in distal-mesial direction.

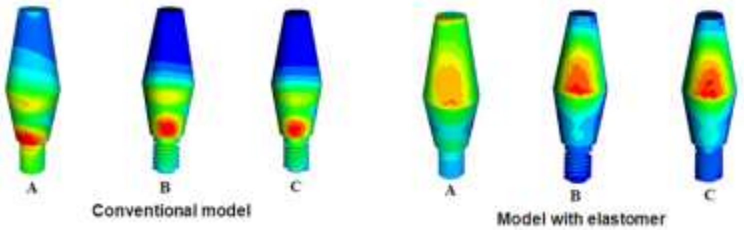


Fig. 6: Distribution of stresses on the abutment in both models
A. Coronal-apical load, B. Lingual-buccal load, C. Distal-mesial load

3.5. Damping Material (Elastomer)

On the elastomer, the maximum von Mises stress distribution showed an increase at the elastomer implant connection area in the buccal region for coronal-apical load, in the buccal–lingual region for buccal–lingual load and in the distal-mesial region for distal mesial load in our daily-life chewing (Fig. 7).

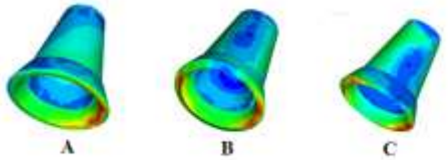


Fig. 7: Distribution of stresses on the elastomer
A. Coronal-apical load, B. Lingual-buccal load, C. Distal-mesial load

3.6. Mandibular bone

In both models the peak of stress values were predominantly found in cortical bone around the cervical region of the implants. For coronal-apical loading the highest stress was concentrated on the buccal bone plates. Due to loading in a lingual-buccal direction, the tensile stress was located on the buccal cortical plate. For distal-mesial loading the maximum stress was generated at the distal bone plate (Fig.8).

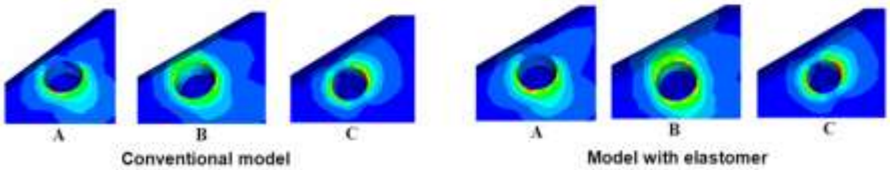
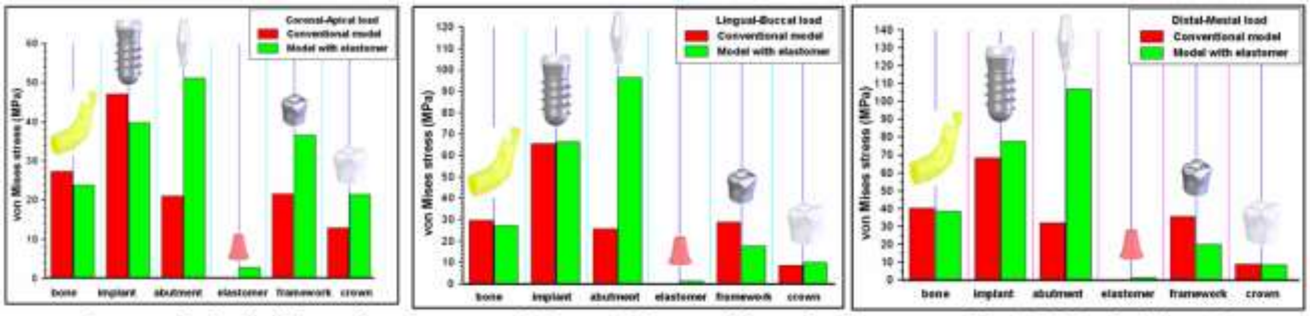


Fig. 8: Distribution of stresses on the mandibular bone in both models
A. Coronal-apical load, B. Lingual-buccal load, C. Distal-mesial load



Coronal-Apical Load Lingual-Buccal Load Distal-Mesial Load

Fig. 9: Histograms of comparison of von Mises stresses for each component in both models under the different load directions.

Irrespective of the model and the loading condition maximum amount of Von Mises stress was found in the cortical bone concentrated at the area adjacent to the implant neck [7-14], and for each model investigated, the highest recorded stresses were those generated by horizontal loads.

In the model with elastomer studied, reduction in the intensity of cervical shearing stresses was measured. Whatever the load orientation, the conventional implant transmitted the highest stresses to the neck region of the implant. In the second model, the elastomer deforms to a large extent and absorbs the load. Under coronal-apical load, stresses decreased by 15.48 % in the implant neck. Under horizontal loads, stresses decreased by 6.92% for lingual-buccal loading and by 14.42 % for distal-mesial loading. This is demonstrated by the decrease of stresses also observed in the cortical bone adjacent to the neck of implant. Under coronal-apical load, stresses decreased by 12.58 % in the cortical bone. Under horizontal loads, stresses decreased by 8.46 % for lingual-buccal loading and by 3.89 % for distal-mesial loading.

4. Discussion

The aim of this study was to predict the stress distributions in the mandibular bone and the different components of dental implants with and without an elastomer. Results of the study are used to investigate the influence of the elastomer material on the load transfer to the implant. All the parameters of both models were kept identical except for the prosthetic design.

Comparative FEM stress analyses between different implant geometries or different implant prosthetic concepts under the same conditions have been previously reported, [6-9] often comparing new geometries to classical implant forms. Comparisons under different modeling conditions could serve as a reference, but do not provide conclusive proof. However, different studies presented comparisons with the Brånemark system, used as a practical standard as it can provide predictable and thoroughly studied clinical results [8-10].

Irrespective of the direction and magnitude of loading, the implant with abutment resist to maximum amount of stress compared to any other component of the model. The probable reason could be associated to its high elastic modulus (E=110,000 MPa), which is 8 times the elastic modulus of cortical bone (E=14500 MPa) and 80 times the elastic modulus of cancellous bone (E=1370 MPa).

In both models, the extreme stresses in the mandibular bone occur in the layer of cortical bone adjacent to the neck of the implants. This is because (1) the surface area between the implant and the cortical bone is much smaller than the surface area between the implant and the cancellous bone, and (2) In addition, the cortical bone is more than 10 times stiffer than the cancellous bone. This suggests that great importance to be attached is 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 [11]. Although results from this study cannot be used to predict clinical performance of the new implant system with elastomer, even though they indicate that also seem to provide an effective solution that does not compromise the stress levels developed in the bone. Clinical decision must not be based solely on a geometry that takes peak stresses away from the bone crest, as stated by Akpinar and associates [12-15].

5. Conclusions

Stress analyses of two different implant geometries, a conventional and a new model with elastomer, were performed, using the finite element method, leading to the following conclusions:-

- Both studied geometries presented quite similar qualitative stress distribution
- Stresses in the new implant system with elastomer 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 use of prosthetic materials with lower stiffness was capable to diminish or to delay the loads transmitted to implants and to the bone.
- Indication found that new model generates high stress induced problems when compared to conventional implant in the abutement compound.

6. References

[1] R. Gassner, J. Vasquez Garcia, W. Leja and M. Stainer, “Traumatic dental injuries and Alpine skiing. Endod Dent Traumatol,” 2000 ; 16:122-127.

http://dx.doi.org/10.1034/j.1600-9657.2000.016003122.x

[2] L. A. Weinberg, “The biomechanics of force distribution in implant-supported prostheses. Int J Oral Maxillofac Implants,”1993;8:19-31.

[3] S. A. Hansson,”Conical implant–abutment interface at the level of the marginal bone improves the distribution of stresses in the supporting bone,”Clin Oral Implants Res, 2003; 14: 286-293.

http://dx.doi.org/10.1034/j.1600-0501.2003.140306.x

[4] S. Ishigaki, T. Nakano, S. Yamada, T. Nakamura and F. Takashima, “Biomechanical stress in bone surrounding an implant under simulated chewing,”Clinical Oral Implants Research, 2003; 14:97-102.

http://dx.doi.org/10.1034/j.1600-0501.2003.140113.x

[5] J.B. Brunski, “In vivo bone response to biomechanical loading at the bone/dental–implant interface,”Adv Dent Res, 1999; 13:99.

http://dx.doi.org/10.1177/08959374990130012301

[6] R. Bassit, H. Lindstrom and B. Rangert, “In vivo registration of force development with ceramic and acrylic resin occlusal materials on implant-supported prostheses,”Int J Oral Maxillofac Implants, 2002;17(1):17-23.

[7] J.P. Geng, Q.S. Ma, W. XU, K.B.C. Tan and G.R. Liu, “ Finite element analysis of four thread-form configurations in a stepped screw implant,”J Oral Rehabil, 2004;31:233–239.

http://dx.doi.org/10.1046/j.0305-182X.2003.01213.x

[8] J.P. Geng, K.B. Tan and G.R. Liu, “Application of finite element analysis in implant dentistry: a review of the literature,” J Prosthet Dent, 2001; 85(6): 585-98.

http://dx.doi.org/10.1067/mpr.2001.115251

[9] H.J. Chun, S.Y. Cheong, J.H. Han, S.J. Heo, J.P. Chung and I.C. Rhyu,“Evaluation of design parameters of osseointegrated dental implants using finite element analysis,” J Oral Rehabil, 2002; 29(6): 565-74.

http://dx.doi.org/10.1046/j.1365-2842.2002.00891.x

[10] O. C. Zienkiewicz,“The Finite Element Method in Engineering Science,” 4th edn. McGraw-Hill, New York, 1989.

[11] R.B. Ashman and W.C. Van Buskirk,“ The elastic properties of a human mandible,” Adv Dent Res, 1987; 1: 64-7.

[12] S. Ishigaki, T. Nakano, S. Yamada, T. Nakamura and F. Takashima,“ Biomechanical stress in bone surrounding an implant under simulated chewing,” Clin Oral Implants Res, 2003; 14: 97-102.
http://dx.doi.org/10.1034/j.1600-0501.2003.140113.x

[13] A. Caglar, C. Aydin, J. Ozen, C. Yilmaz and T. Korkmaz,“ Effects of mesiodistal inclination of implants on stress distribution in implant-supported fixed prostheses,” Int J Oral Maxillofac Implants, 2006; 21: 36-37.

[14] H. Van Oosterwyck, J. Duyck, J. Vander Sloten, G. Van Der Perr and I. Naert,“ Peri-implant bone tissue strains in cases of dehiscence: a finite element study,” Clin Oral Implants Res, 2002; 13: 327-33.
http://dx.doi.org/10.1034/j.1600-0501.2002.130314.x

[15] C. Lídia, A. Ramos and A. Simões, “Finite element analysis of a dental implant system with an elastomeric stress barrier,”Summer bioengineering conference, June 25–29, Sonesta Beach Resort in Key Biscayne, Florida, 2003.