

Stress Distribution Around Maxillary Anterior Implants as a Factor of Labial Bone Thickness and Occlusal Load Angles: A 3-Dimensional Finite Element Analysis

Marzieh Alikhasi, DDS, MS¹
 Hakimeh Siadat, DDS, MS²
 Allahyar Geramy, DDS, MS³
 Ahmad Hassan-Ahangari, DDS, MS^{4*}

The purpose of this study was to evaluate the influence of the stress/strain distribution in buccal bone of an anterior maxillary implant using 3 bone thicknesses under 5 different loading angles. Different testing conditions incorporating 3 buccal bone thicknesses, 3 bone compositions, and 5 loading angles of an anterior maxillary implant were applied in order to investigate the resultant stress/strain distribution with finite element analysis. The maximum equivalent stress/strain increased with the decreasing of loading angle relative to the long axis. In addition to loading angle, bone quality and quantity also influenced resultant stress distribution. Dental practitioners should consider combinations of bone composition, diameter, and load angulations to predict success or failure for a given implant length and diameter.

Key Words: dental implants, bone quality, biomechanics, finite element analysis, off-axis load

INTRODUCTION

There are several biomechanical factors including type of loading, implant geometry, surface structure, quality and quantity of the surrounding bone, and the nature of the bone-implant interface that could affect stresses and strains around osseointegrated dental implants.¹ The existing 3-dimensional bone not only affects the ideal implant position but also influences the degree of bone remodeling following implant placement.^{2,3} Thick-

ness of the bone in the bucco-oral and mesiodistal of the implant is an important factor that affects resorption, which leads to soft-tissue shrinkage or edema.³ There is insufficient evidence to set a threshold for minimal buccal bone thickness to ensure an optimal esthetic outcome, although it has been suggested that it is crucial to have a buccal bone plate of at least 1 mm.^{4,5} Buser et al² demonstrated several clinical guidelines regarding the correct implant positioning in relation to bucco-oral bone dimensions especially in the maxillary anterior zone. They have recommended that the bucco-oral bone thickness should be at least 2 mm around the implant.² This buccal bone thickness was advocated to ensure proper soft-tissue support, avoid resorption of the facial bone wall following restoration, and minimize the risk for peri-implant soft-tissue recessions.^{6,7} Since the occlusal loads of maxillary anterior teeth are in the outward direction, this buccal bone has a special role in enduring the loads. When the load direction is not in the long axis of an implant, the buccal, lingual, and proximal

¹ Dental Research Center and Department of Prosthodontics, School of Dentistry, Tehran University of Medical Sciences, Tehran, Iran.

² Dental Implant Research Center and Department of Prosthodontics, School of Dentistry, Tehran University of Medical Sciences, Tehran, Iran.

³ Department of Orthodontics and Dental Research Center, School of Dentistry, Tehran University of Medical Sciences, Tehran, Iran.

⁴ Department of Prosthodontics, School of Dentistry, Shiraz University of Medical Sciences, Shiraz, Iran.

* Corresponding author, e-mail: gueramy@tums.ac.ir
 DOI: 10.1563/AAID-JOI-D-10-00198

bone thicknesses around a dental implant endure different forces.⁸ In addition to bone thickness and load direction, the quality of the available bone as stated by Zarb and Schmitt is a determining factor in treatment planning, implant design, surgical approach, and healing time.⁹⁻¹¹ Therefore, when a clinician decides to insert an implant in the anterior maxillary section, he or she should consider all of these factors to reduce stress concentration around the implant.

One of the useful methods to assess different mechanical parameters in living tissues is finite element analysis (FEA), which has shown its efficiencies in various situations.¹²⁻²⁰ Some investigators studied the influence of different loading angles on stress concentration in the bone around the implant and indicated that the loading was a significant factor influencing the stress created in the bone.^{4,8,21,22} Others studied the influence of bone quality and quantity on stress concentration.²³⁻²⁵ However, there is no study to show which of these factors have greater influence on stress and strain fields around the dental implant. Finite element analysis was used in the present study to examine the influence of the thickness of bucco-oral bone and composition of it in crestal bone strains around an anterior maxillary implant using 5 off-axis loads.

MATERIALS AND METHODS

Nine different models of the anterior part of the maxilla were designed. Each model contained an implant, its abutment, cancellous bone, cortical bone, and gingiva. The difference between the first 3 models was in the labial bone thickness (the least distance between the implant fixture and the labial part of the cortical bone in the midline at the level of the first thread), which was 1.5 mm in its normal situation. The thickness was 0.75 mm and 2.0 mm in other models. The other models had 1.5 mm of labial bone thickness and different bone compositions. The normal situation was defined as having 0.75 mm of cortical bone and the same thickness of the cancellous bone in the labial bone. The fourth and fifth models had 0.2 mm of cortical bone and 1.3 mm of cortical bone, respectively. In the last phase of the analysis, the angle of crown loading was changed between 15° to 60° to the crown surface (models sixth to ninth; Figure 1). The models

were designed in SolidWorks 2006 (Concord, Mass) and then transferred to ANSYS Workbench Version 11 (Southpointe, Cononsburg, Penn) for the solving process. Material properties were defined according to previous studies (Table 1). A convergence test was done to find the best element size. Meshing was done by the powerful meshing program in Workbench with 0.2 mm of element sizing. Meshed models contained 83 248 nodes and 64 115 elements (47 727 body elements and 16 388 contact elements) with a small difference in various models (Figure 2). The base and lateral parts of each model were restrained. The palatal side of the abutment was loaded by a 178-N force perpendicular to the palatal surface in each model.^{21,22} The mechanical properties of the materials used were considered as presented in Table 1. The von Mises stress along a path was assessed. The defined path started from the crestal bone on the labial side and ended vertically in a point corresponding to the apical part of the implant in the midline of the labial bone surface.

RESULTS

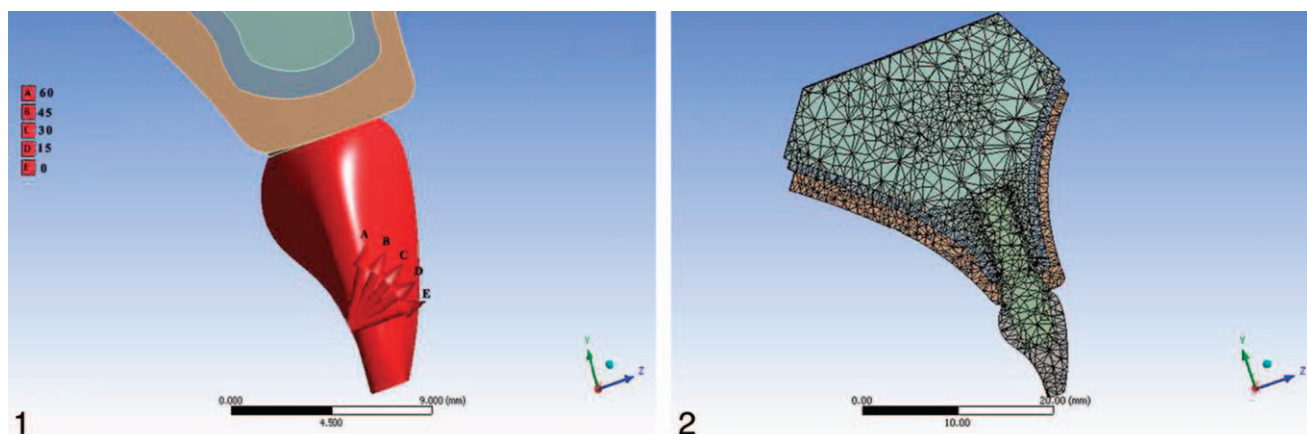
The results were divided into 3 parts and are presented in Table 2.

Different labial bone thicknesses

Numeric findings for the different labial bone thickness revealed the highest findings in the crestal bone node adjacent to the implant and decreased toward the apical node. These findings were 154.16 MPa in crestal node and 2.96 MPa in the apical area of the model with 1.5 mm labial bone thickness (model 1). The crestal bone stress in the model with 0.75 mm labial bone thickness (model 2) was 197.47 MPa and decreased to 2.83 MPa in its end node. In the third model with 2.0 mm labial bone thickness, the highest von Mises stress was 127.25 MPa, which decreased to 2.92 MPa in its end node. Figure 3 describes the von Mises stress around the first to third threads of the implant along the labial surface of different bone thickness.

Different labial bone composition

The maximum recorded stress was in the crestal bone node adjacent to the implant with 0.75 mm of cortical and cancellous bone (model 1) and started a



FIGURES 1 AND 2. FIGURE 1. Load angulations in different models. FIGURE 2. Sagittal view of the finite element model.

gradual decrease toward the apical node in all bone composition models. In this model, the maximum von Mises stress was 150.51 MPa, which decreased to 3.36 MPa in its end node. Decreasing the cancellous bone thickness in the fifth model (0.2 mm cancellous bone) revealed 139.6 MPa as the crestal bone stress and 2.44 MPa in its apical part node. In the fourth model with decreased cortical bone thickness (0.2 mm cortical bone), the highest von Mises stress was 143.72 MPa, which decreased to 3.25 MPa in its end node (Figure 4).

Different load angles

The highest von Mises stresses that were recorded in all models were in the crestal bone. Among these models, the maximum stress was found in the sixth model (loading angle of 15°). Decreasing the loading angle was corresponding to an increase in the stress findings in the crestal and other nodes. In the sixth model the, highest finding was 156.67 MPa, which decreased to 2.3 MPa in its end node. Increasing the load angle to 30° (model 7) caused a stress of 139.69 MPa in the crestal bone and 2.3 MPa in its end node. When the load angle reached 45° (model 8), the highest von Mises stress was 119.92

MPa, which decreased to 2.1 MPa in its end node. Since the load angle became closer to the long axis of teeth (model 9), the stresses decreased, and the least amount of stresses (72.96 MPa at crest and 1.84 MPa in its end node) were recorded (Figure 5).

DISCUSSION

A thorough investigation of the fields of stress and strain in an implant/jawbone system is of vital importance to protect the jawbone and to minimize implant failure. Several investigations have demonstrated that the diameter and length of an implant have significant effects on stress/strain fields in the bone and implant.^{7,19} An optimized design of an implant in combination with a carefully controlled implant insertion could also be effective in improving the biomechanical environment for the maintenance of bone in implant/bone systems. Other factors that clinicians must consider during implant insertion are the thickness of the buccal bony wall of the implant prepared site and angulated positioning of the implant. An implant, when inserted into jawbone with a reduced buccal bone thickness and loaded with an oblique loading angle, seems to be most unfavorable for stress distribution in both bone and implant. However, clinical trial studies that could compare different angulations, bone thicknesses, and compositions are difficult to design. This article explored these factors with FEA to assess the critical buccal bone dimension and composition around implants for an optimal stress/strain outcome.

As most bone loss is initiated at the alveolar crest around an implant neck,²⁰ this study focused

	Young's Modulus (MPa)	Poisson's Ratio
Porcelain	70 000	0.22
Titanium	96 000	0.36
Cancellous bone	13 400	0.38
Cortical bone	34 000	0.26
Gingiva	19.6	0.3

TABLE 2

Principle stresses (MPa) arising in the bone around the implant under different conditions

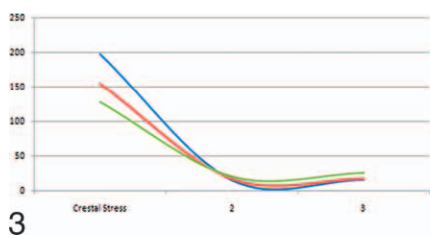
Model	Cancellous Bone Diameter	Cortical Bone Diameter	Full Diameter of Bone	Load Angulation	Stress	Bank
2	0	0.75	0.75	0	197.47	1
6	0.75	0.75	1.5	15	156.67	2
1	0.76	0.75	1.5	0	150.51	3
4	1.3	0.2	1.5	0	143.72	4
5	0.2	1.3	1.5	0	139.6	5
7	0.75	0.75	1.5	30	139.69	6
3	0.75	1.25	2	0	127.25	7
8	0.75	0.75	1.5	45	119.92	8
9	0.75	0.75	1.5	60	72.96	9

on the values of von Mises stress and von Mises strain on the buccal crestal bone for all variations. In the present study, similar to the previous studies,⁶⁻⁸ it was found that elevated stress levels were located at the crestal bone in all combinations of the different parameters. In addition, reducing buccal bone thickness increased the strain distribution pattern and its magnitude in both bone layers. Moreover, the angle of force application affected the stress magnitude and distribution pattern in the bone. Whenever the load direction was closer to the implant long axis, the stress values were reduced and distributed more symmetrically in the bone, which agrees with the findings of the study by Qian et al.⁸

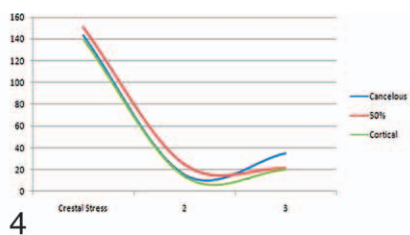
Among the 9 models evaluated in this study, the maximum stress value (197.47) was observed in the second model, in which the buccal bone thickness was 0.75 mm (Table 2). Teughels et al,⁵ in a systematic review, suggested that a buccal bone thickness of about 2 mm would reduce the incidence and amount of vertical bone loss. However, they could not find the exact threshold of bucco-oral bone dimension to achieve the best results in terms of esthetic and stress distribution.⁵ This suggests the need to design further studies

that can simulate various bucco-oral bone thicknesses. Also, it could be valuable to determine the threshold buccal bone around various designs on implants.

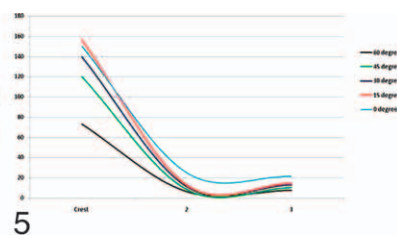
Bone quality is another factor that has been shown to influence the strains/stresses around endosseous implants.^{6,23} The present analysis considered various bone compositions. In general, good agreement was found with previous research in that an increase in cortical bone was associated with reductions in stress (models 4 and 5). This finding should be considered in bone-grafting procedures in which buccal bone is increased, although the composition is mainly cancellous. However, this study demonstrated that as the diameter of either cancellous or cortical bone increased, their ability to carry the load increased. It means that when there is 0.2 mm cortical bone, increasing cancellous bone from 0.75 to 1.3 reduced stresses more compared with an equal diameter of 0.75 of cancellous and cortical bone. An explanation of this finding is that cortical bone is typically anisotropic and its ultimate stress is higher in compression.²⁶ However, the strength of the trabecular bone has been reported to be the same in tension and compression.²⁶ Therefore, when the



3



4



5

FIGURES 3–5. FIGURE 3. The von Mises stress along the labial surface of different bone thicknesses (first to third threads). FIGURE 4. The von Mises stress along the labial surface of different bone compositions (first to third threads). FIGURE 5. The von Mises stress along the labial bone surface of different load angulations (first to third threads).

load direction is not along the long axis of the implant (as shown in this study), both trabecular and cortical bone seem to be necessary.

There are inherent limitations in this FEA that limit extrapolating the results to clinical situations. The structures in the model were all assumed to be homogenous and isotropic and to possess linear elasticity. All interfaces between the materials were assumed to be bonded or osseointegrated, and cement thickness layer was also ignored. Thus, since 3-dimensional (3D) bone stress is a complicated function of various parameters, further well-designed 3D studies are still needed to address this issue more clearly. However, the outcome of this study within its limitations will help dental practitioners to predict success or failure for combinations of bone composition, thickness, and load angulation with a given implant length and diameter. Moreover, the findings of the analysis may be translated and used by the clinician in the decision-making process regarding the positioning of implants.

ABBREVIATIONS

3D: 3-dimensional
FEA: finite element analysis

REFERENCES

- Geng JP, Tan KB, Liu GR. Application of finite element analysis in implant dentistry: a review of the literature. *J Prosthet Dent.* 2001;85:585–598.
- Buser D, Martin W, Belser UC. Optimizing esthetics for implant restorations in the anterior maxilla: anatomic and surgical considerations. *Int J Oral Maxillofac Implants.* 2004;19:43–61.
- Esposito M, Ekstubb A, Gröndahl K. Radiological evaluation of marginal bone loss at tooth surfaces facing single Brånemark implants. *Clin Oral Implants Res.* 1993;4:151–157.
- Grunder U, Gracis S, Capelli M. Influence of the 3-D bone-to-implant relationship on esthetics. *Int J Periodont Restor Dent.* 2005;25:113–119.
- Teughels W, Merheb J, Quirynen M. Critical horizontal dimensions of interproximal and buccal bone around implants for optimal aesthetic outcomes: a systematic review. *Clin Oral Implants Res.* 2009;20(suppl 4):134–145.
- Holmes DC, Loftus JT. Influence of bone quality on stress distribution for endosseous implants. *J Oral Implantol.* 1997;23:104–111.
- Petrie CS, Williams JL. Comparative evaluation of implant designs: influence of diameter, length, and taper on strains in the alveolar crest: a three-dimensional finite-element analysis. *Clin Oral Implants Res.* 2005;16:486–494.
- Qian L, Todo M, Matsushita Y, Koyano K. Effects of implant diameter, insertion depth, and loading angle on stress/strain fields in implant/jawbone systems: finite element analysis. *Int J Oral Maxillofac Implants.* 2009;24:877–886.
- Cochran DL. The scientific basis for and clinical experiences with Straumann implants including the ITI dental implant system: a consensus report. *Clin Oral Implants Res.* 2000;11:33–58.
- Quirynen M, Naert I, van Steenberghe D. Fixture design and overload influence marginal bone loss and fixture success in the Branemark system. *Clin Oral Implants Res.* 1992;3:104–111.
- Zarb GA, Schmitt A. Implant prosthodontic treatment options for the edentulous patient. *J Oral Rehabil.* 1995;22:661–671.
- Korioth TW, Versluis A. Modeling the mechanical behavior of the jaws and their related structures by finite element (FE) analysis. *Crit Rev Oral Biol Med.* 1997;8:90–104.
- Geramy A. Alveolar bone resorption and the center of resistance modification (3-D analysis by means of the finite element method). *Am J Orthod Dentofacial Orthop.* 2000;117:399–405.
- Geramy A. Initial stress produced in the periodontal membrane by orthodontic loads in the presence of varying loss of alveolar bone: a three-dimensional finite element analysis. *Eur J Orthod.* 2002;24:21–33.
- Geramy A. Optimization of unilateral over-jet management: three-dimensional analysis by the finite element method. *Angle Orthod.* 2002;72:585–592.
- Geramy A, Morgano SM. Finite element analysis of three designs of an implant-supported molar crown. *J Prosthet Dent.* 2004;92:434–440.
- Geramy A, Sharafoddin F. Abfraction: 3D analysis by means of the finite element method. *Quintessence Int.* 2003;34:526–533.
- Geramy A, Faghihi S. Secondary trauma from occlusion: three-dimensional analysis using the finite element method. *Quintessence Int.* 2004;35:835–843.
- Petrie CS, Williams JL. Comparative evaluation of implant designs: influence of diameter, length, and taper on strains in the alveolar crest: a three-dimensional finite-element analysis. *Clin Oral Implants Res.* 2005;16:486–494.
- Stegaroiu R, Sato T, Kusakari H, Miyakawa O. Influence of restoration type on stress distribution in bone around implants: a three-dimensional finite element analysis. *Int J Oral Maxillofac Implants.* 1998;13:82–90.
- Hsu ML, Chen FC, Kao HC, Cheng CK. Influence of off-axis loading of an anterior maxillary implant: a 3-dimensional finite element analysis. *Int J Oral Maxillofac Implants.* 2007;22:301–309.
- Clelland NL, Lee JK, Bimbenet OC, Branthy WA. A three-dimensional finite element stress analysis of angled abutments for an implant place in the anterior maxilla. *J Prosthodont.* 1995;4:95–100.
- 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. *Int J Oral Maxillofac Implants.* 2003;18:357–368.
- Okumura N, Stegaroiu R, Kitamura E, Kurokawa K, Nomura S. Influence of maxillary cortical bone thickness, implant design and implant diameter on stress around implants: a three-dimensional finite element analysis. *J Prosthodont Res.* 2010;54:133–142.
- 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 Prosthet Dent.* 2005;93:38–44.
- Martin RB, Burr DB, Sharkey NA. *Skeletal Tissue Mechanics.* 1st ed. New York: Springer; 1998:127–178.