

Bone Response From a Dynamic Stimulus on a One-Piece and Multi-Piece Implant Abutment and Crown by Finite Element Analysis

Habib Hajimiragha, DDS¹
 Mohammadreza Abolbashari, DDS^{2*}
 Saeed Nokar, DDS²
 AmirHossein Abolbashari, DDS³
 Mehrdad Abolbashari, DDS⁴

The present study was done to evaluate the effects of different types of abutments on the rate and distribution of stress on the bone surrounding the implant by dynamic finite element analysis method. In this study two ITI abutment models—one-piece and multi-piece—along with fixture, bone, and superstructure have been simulated with the help of company-made models. The maximum Von Mises stress (MVMS) was observed in the distobuccal area of the cortical bone near the crest of implant in two implant models. In the multi-piece abutment, MVMS was higher than the one-piece model (27.9 MPa and 23.3 MPa, respectively). Based on the results of this study, it can be concluded that type of abutment influences the stress distribution in the area surrounding the implant during dynamic loading.

Key Words: *finite element analysis, one-piece abutment, multi-piece abutment, implant, stress*

INTRODUCTION

Quality and quantity of bone are important in implant dentistry. They imply the characteristics of the external surface of the edentulous area, which must be considered for implants. Moreover, bone has an internal structure that is described as “quality” or “density,” which represent the bone strength.¹ The mechanical distribution of stress happens first at the contact area of the implant and the bone.² The percentage

of bone-implant contact in the cortical bone is noticeably more than in the trabecular bone.² Researchers have studied the effect of implant design on stress concentration on the bone during loading; findings have shown that implant design is an important factor in stress distribution on bone.¹ The mechanical stress causes bone strain, which means deformation of the bone. These bone deformations in their length are called “microstrains.” The amount of strain depends directly on the applied stress on the bone through implant and the mechanical characteristics of the bone (eg, stiffness).³

The stiffness of a titanium implant and its alloys is relatively high on the cortical bone. When the implant is under loading, the stress will be transmitted to the bone and is maximal at the coronal area of the bone. This is consistent with the general mechanical rule that says when two parts are in direct contact and one part is loaded, the highest stress will be on the first contact point

¹ Department of Fixed Prosthodontics and Occlusion, Tehran University of Medical Sciences, Tehran, Iran.

² Department of Fixed Prosthodontics and Occlusion, International Campus of Tehran University of Medical Sciences, Tehran, Iran.

³ Islamic Azad University, Dental Branch of Tehran, Tehran, Iran.

⁴ Center for Optoelectronics and Optical Communications, Center for Precision Metrology, University of North Carolina, Charlotte, NC.

* Corresponding author, e-mail: maalcomxx@yahoo.com

DOI: 10.1563/AAID-JOI-D-10-00170

between the two.³ This principle was observed in a simulation of the loaded implant in the photoelastic method and three-dimensional finite element analysis.³

Mechanical failures, such as loosened or fractured occlusal screw or loosened abutment screws, are related to the implant-abutment connection interface.⁴ There are two types of connections commonly used for securing the abutment to the implant⁴: one screw and a tapered interface fit (Morse taper). For the systems in which screws are used, the implant-abutment connection depends on preload force, which is predetermined. The tapered interface fit depends on high connection pressure and resultant frictional resistance in the implant-abutment connection interface to create a reliable connection.⁴

One-piece abutments are often selected for several splinted implants.⁵ A one-piece abutment includes one or two flat surfaces, which limits casting movement and puts the cement under pressing force. There is no need to create tension in the screw as the splinted crown prevents the abutment from being rotated. The multi-piece abutment is more resistant to fracture than is the one-piece abutment since the annulus of the implant body is placed lower. In a multi-piece abutment, an anti-rotating part surrounds the implant body while the abutment screw connects the abutment to the implant body.⁵

One of the important factors in stress distribution on the bone surrounding the implant is the abutment design, which we have taken into consideration in this study. Chun et al⁶ have done research on the influence of the different types of abutments (one-piece, internal hexagon and external hexagon implants) as relates to stress distribution on the bone under various loading conditions using finite element analysis. They showed that abutment type had significant influence on the stress distribution in bone due to different load transmission mechanisms and differences in the size of implant-abutment connection interface.

Hansson⁷ studied the stress distribution pattern on the bone in conical and flat-top abutment-implant contact interfaces. The study revealed that the axial loading causes a deeper and more uniform stress on the implant surface with conical contact area and adjacent bone, while the stress concentration in an implant with flat-top contact area was

observed in the most coronal part of the bone margin.

Most stress analyses of implant-supported prostheses have been done using static finite element analysis methods. Although the static response of implants is well known, the dynamic responses of dental implants are not yet quite known. There are few resources about the dynamic finite element analysis in evaluating dental implants stress. Williams et al⁸ investigated the dynamic responses of dental implants with the impulse excitation methods, and they proved that the implant responses depend on duration and direction of the impulses.⁸

The finite element model (FEM) is one of the best tools for simulation of dental restorations under loading. The information that FEM provides usually cannot be obtained from experiments. Moreover, the precise prediction of dental implant stability and failure mechanisms using FEM will be helpful in reducing the number of clinical experiments.⁹ A study by Huang et al,¹⁰ has assessed the resonance frequency to determine dental implants' vibrating behavior under a variety of surrounding bone conditions. However, the authors did not analyze the dynamic response of dental implants under the action of any general time-dependent load. Transient dynamic analysis was performed to characterize the dynamic response of a structure under the action of any general time-dependent load.

To the best of our knowledge, there has been no 3D dynamic finite element analysis on dynamic responses in dental implants with different abutments. However, there are several studies on stress analysis around implants with different abutment connections using static finite element methods.^{3,4,6,11-13} Evaluating the influence of an abutment type is difficult with static methods. Dynamic analysis is more appropriate for evaluation of the effect of implant-abutment connection interface design in implant systems. In this current study, the effects of abutment type on the distribution and amount of stress on the bone surrounding the implant were investigated by dynamic finite element analysis.

MATERIALS AND METHODS

This section includes measurement, modeling, and applying specifications to the model.

Measurement

For the measurement, it was necessary to measure parts for the simulation. The components used in this research include:

1. Implant fixture, 4.1×12 mm standard ITI. Internal diameter of fixture in platform area is 3.5 mm, and it is thinned with an internal slope of 8 degrees. There is a regular octagon with the length of 1.25 mm in the interior part of the implant. An annulus is designed in the depth of 3.4 mm from the implant platform area.
2. The solid abutment ITI was considered the one-piece abutment. The specification of this abutment includes a flat surface on the abutment with a screw with five standard threads that has apical extension. Since the screw is not an independent part in this abutment, it is called a "one-piece abutment." This abutment is used with regular neck (RN) and wide neck platforms. Cemented crown and bridge restorations are used on solid abutments.
3. The other abutment that was used in this study was a synOcta abutment ITI (Institut Straumann AG, Basel, Switzerland), which is used with regular and wide neck platform implants. It is possible to use cemented crown or bridge restorations on this abutment. We used a regular neck synOcta 1.5 screw retained abutment. Since the screw is independent from the abutment and other components are added—gold coping, UCLA cast to abutment, occlusal prosthetic screw—it is considered a "multi-piece abutment." The abutment is a tapered and regular octagon. The abutment screw is standard and has five threads.
4. Gold coping (4.25 mm) and UCLA cast to abutment for multi-piece abutments.
5. Occlusal prosthetic screw.

Two instruments were used for measurement: a microscope and a profile projector with accuracies of 0.005 mm and 0.01 mm, respectively. After measuring the parts, the components were drawn in AutoCAD software as 2D drawings.

Modeling

There are three steps in finite element analysis: preprocessing, processing, and postprocessing.¹⁴

Therefore, after measurement, the next step is modeling. We used SolidWorks 2006 SP0 (DS

SolidWorks, Waltham, Mass) software was used, which is one of the best 3D modeling software available. Figures 1 through 3 display some of the 3D models. Modeling the upper jaw bone in the second premolar area was done according to the study by Koca and colleagues.¹⁵ In this model, the bone was modeled as a trapezoidal block. The modeled bone type was assumed to be a D3 type, according to the Lekholm and Zarb classification.¹⁶ In bone modeling, the dimensions were as follows: 15 mm height, the bone surrounding the implant 0.5 mm in buccolingual direction and 1.5 mm in mesiodistal direction.⁵ The designed bone included cortical bone with 1 mm thickness, which was surrounding the cancellous (trabecular) bone.¹²

For modeling the superstructure, the dimensions of the second premolar tooth of the upper jaw were first extracted according to the Wheeler's tooth anatomy book.¹⁷ The 3DS Max8 software (Autodesk, San Francisco, Calif) was used for the simulation (Figure 4). The porcelain thickness and the coping thickness were considered 1 mm and 0.35 mm, respectively, in the occlusal part.¹⁵

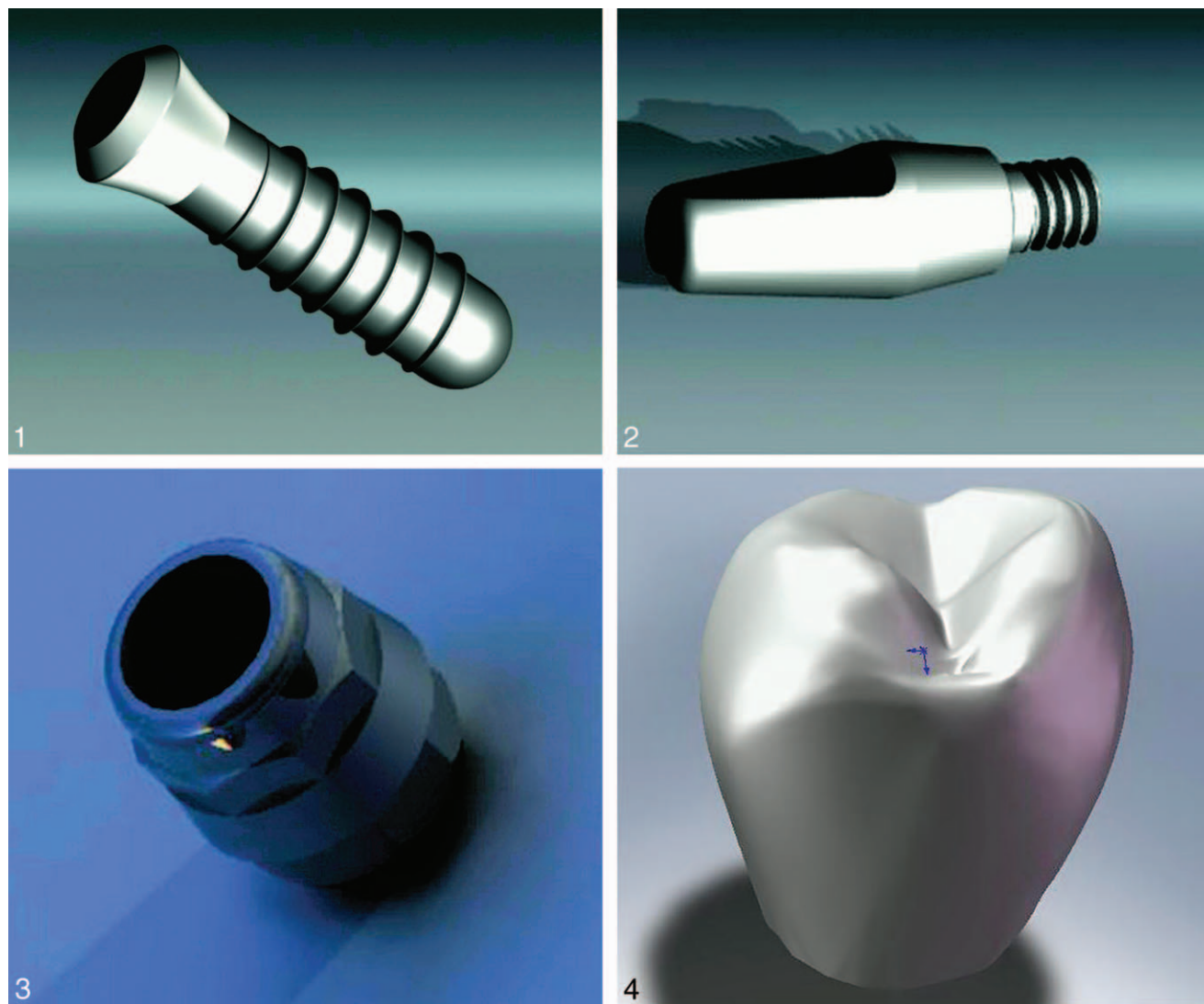
After modeling the parts, bone, and superstructure, the next step was to put the parts together and create the desired model. In SolidWorks, it is called "assembling."

Our study included two models: The first model was based on the one-piece solid abutment RN (regular neck). In this model, the implant fixture was first inserted in the bone. Then, the solid abutment was inserted into the implant fixture and screwed into the annulus of implant body. Finally, the superstructure was placed onto the solid abutment (Figure 5).

In the multi-piece abutment model, the fixture was first inserted in the bone. Then the 1.5 mm synOcta abutment was placed into the implant fixture, and the abutment screw was tightened in the annulus of the implant body. The superstructure was then inserted on the multi-piece abutment. Finally, the occlusal prosthetic screw was screwed onto the multi-piece abutment through the hole in the occlusal side of crown (Figure 6).

Assigning and applying the model specifications

Ansys 10 software (ANSYS, Canonsburg, Pa) was utilized as the finite element analysis software. For analyzing the models, a Pentium dual core with Windows XP operating system was used. First, the



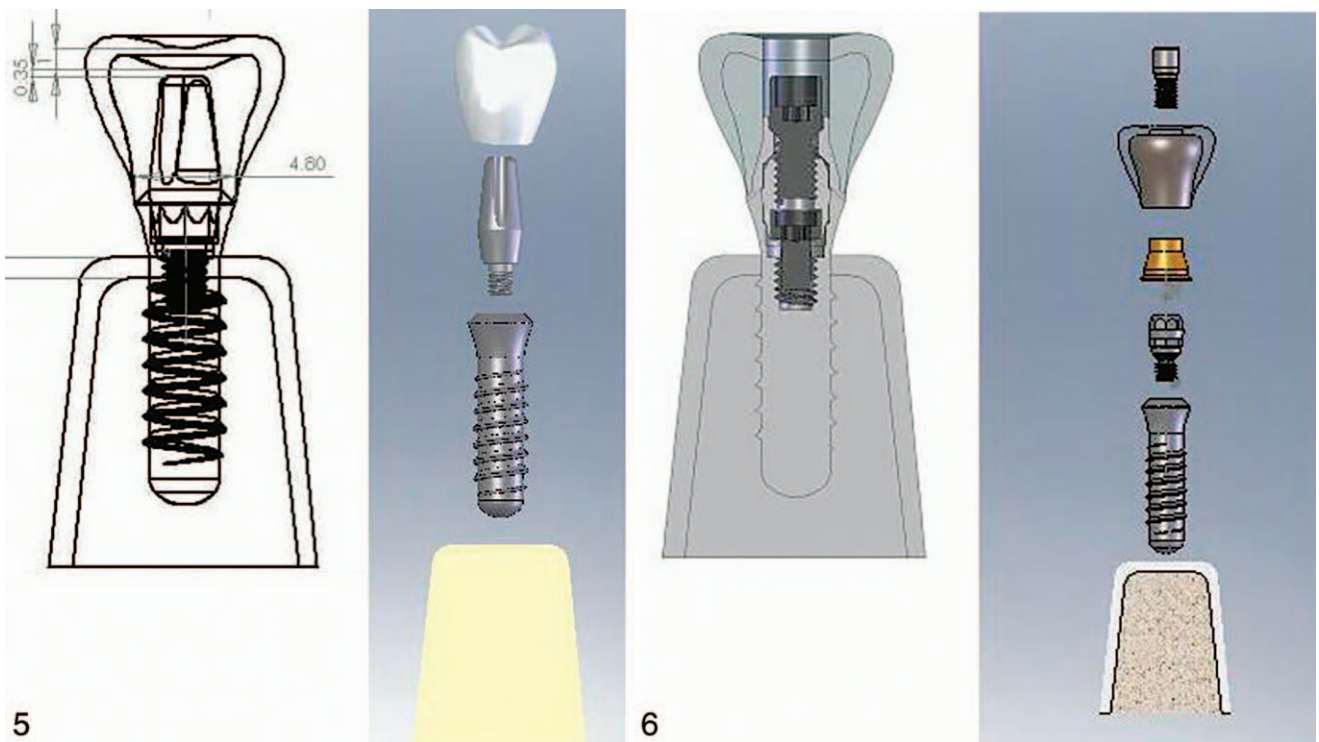
FIGURES 1–4. **FIGURE 1.** 3D model of implant fixture, 4.1 × 12 mm standard ITI. **FIGURE 2.** 3D model of solid abutment ITI (regular neck [RN]) with the length of 5.5 mm. **FIGURE 3.** 3D model of synOcta abutment ITI (RN) for screwed crowns and bridges with the length of 1.5 mm. **FIGURE 4.** Designed superstructure model in second premolar of upper jaw.

physical properties were assigned to the materials. In this study, all the materials were considered as isotropic and homogenous.

The fixture implant and the abutments were pure titanium (grade 4) and the coping and framework were a type of gold alloy called Ceramicor (Cendres+Métaux SA, Biel/Bienne, Switzerland). For dynamic analysis, some specifications of materials (such as density, modulus of elasticity, and Poisson’s ratio) are needed. The specifications for these materials and for the porcelain, cortical bone, and cancellous bone are given in Table 1. The connection between the cortical bone and cancellous bone was considered to be a merged connection.

Osseointegration between the bone and fixture was considered to be 100%.^{1,15} The implant-abutment connection interface was simulated as an interface with 0.5 friction coefficient.⁹ In the multi-piece abutment model, the connection area between the abutment body and the abutment screw was considered a contact joint, as was the interface between the coping and fixture implant.⁹

We used two references to determine the location and number of loading areas, and the angle of loading: Misch¹⁸ and Koca et al.¹⁵ According to these references, the loading zones were assigned to be in two areas of 2–3 mm² including Mesial Fossa and Palatal Cusp crown in the right premolar area of the upper jaw. Two 150 N



FIGURES 5 AND 6. **FIGURE 5.** Assembled schematic model of one-piece abutment solid ITI regular neck. **FIGURE 6.** Assembled schematic model of multi-piece abutment synocta ITI.

loads with a 30-degree angle were assigned for both zones.^{5,15} For the loading profile in dynamic analysis, one cycle of chewing impulse with 0.2-second duration was considered.

For tightening the screw of the multi-piece abutment and one-piece abutment, the company (Straumann AG) suggestion of 35 Ncm torque was used. Torque of 15 Ncm was used for the occlusal prosthetic screw (Table 2).

The last step was meshing these models. With the help of geometric models, the finite element models were simulated by Finite Element Model Builder (Livermore Software Technology, Livermore,

Calif); that is, the geometric models were converted to a set of elements and nodes. In our models, there were some complicated surfaces with very small dimensions that prevented automatic meshing. Therefore, it was done manually, that is, the nodes were made manually and by choosing them in a regular manner, each element was made. The surfaces in contact were made in such a way that the elements matched. In next step, adjusting the loading and the contact surfaces, it was necessary to choose some nodes to be used as the contact surface or the loading surface. These nodes are called the "node set."

TABLE 1

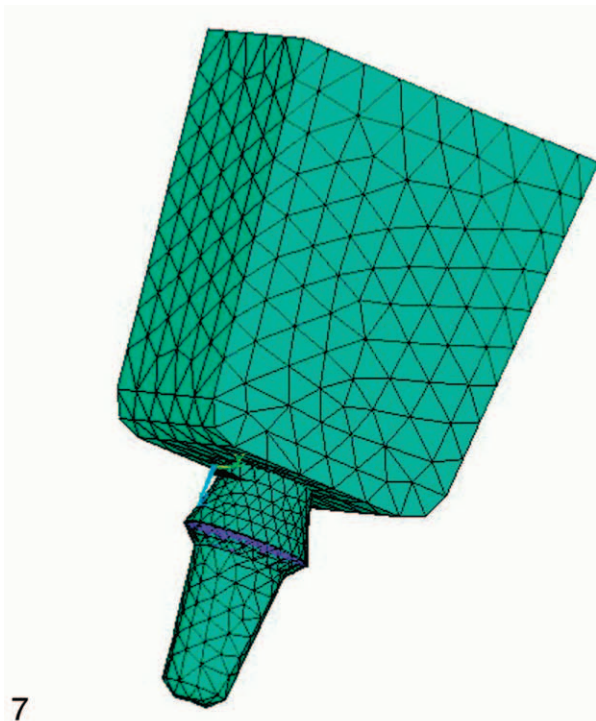
Mechanical properties of materials used in finite element analysis

	Density (g/cm ³)	Modulus of elasticity (MPa)	Poisson's ratio
Porcelain (35)	2.4	68 900	0.28
Gold alloy	17.5	136 000	0.33
Cortical bone (35)	1.3	13 400	0.3
Spongy (cancellous) bone (35)	1.3	1370	0.3
Pure titanium (1)	4.6	110 000	0.33

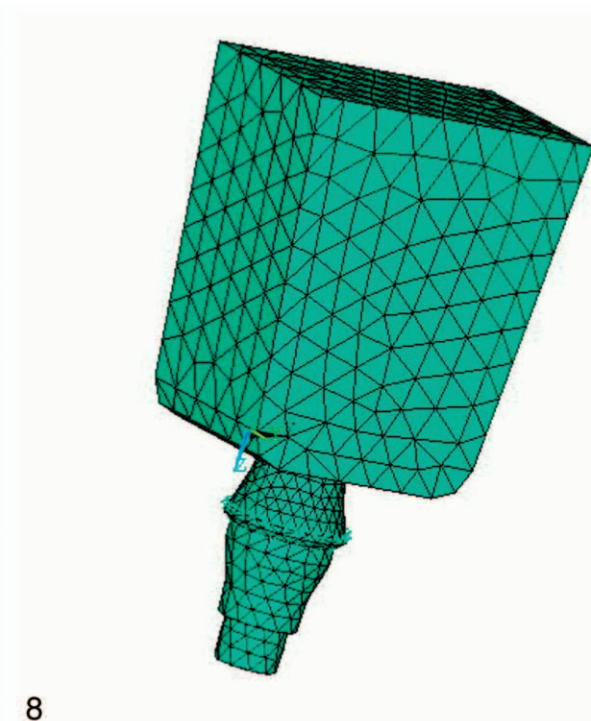
TABLE 2

Preload force on abutment screw equivalent to torque recommended by company (Straumann AG)

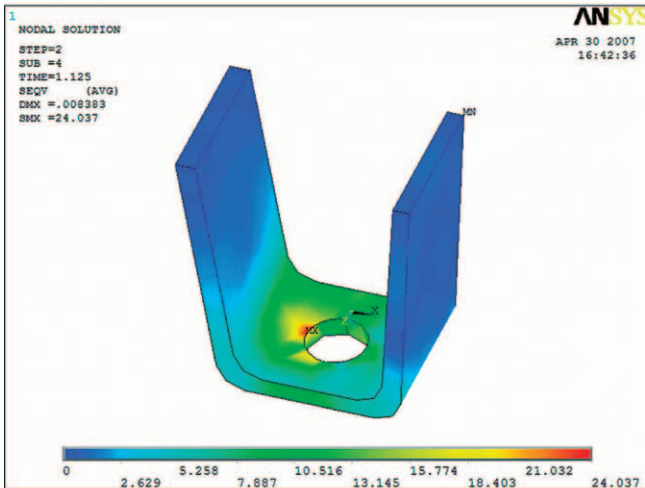
Abutment type	One-piece	Multi-piece	
		Abutment screw	Occlusal prosthetic screw
Recommended torque	35 N.Cm	35 N.Cm	15 N.Cm
Preload force	196 N	196 N	81.5 N



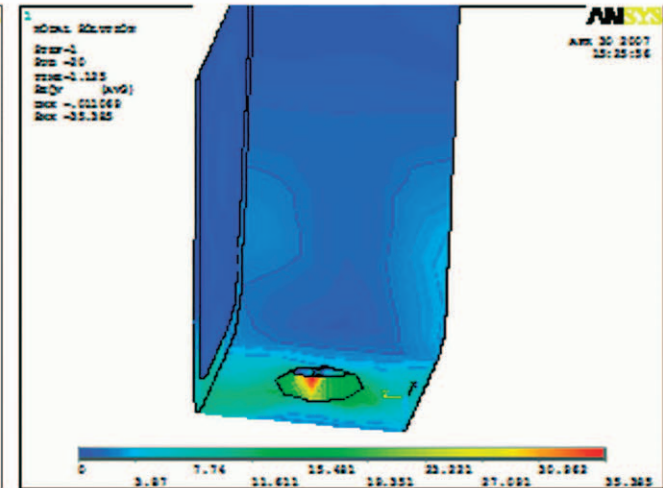
7



8



9



10

FIGURES 7–10. **FIGURE 7.** 3D mesh for one-piece abutment set with surrounding bone. **FIGURE 8.** 3D mesh for multi-piece abutment set with surrounding bone. **FIGURE 9.** Distribution and amount of stress in one-piece abutment set. **FIGURE 10.** Distribution and amount of stress in multi-piece abutment set.

After exporting these models to Ansys, they were reread. The basic selected element in the study was the tetrahedral 45 solid, which included 4 nodes. Figures 7 and 8 demonstrate the meshed model in two models of one-piece abutment and multi-piece abutment.

The one-piece abutment model had 40 338 elements and 14 000 nodes; solving this model took 15 hours. The multi-piece abutment model had 31 500 elements and 8120 nodes; its solving

time was also 15 hours. For processing the models, dynamic analysis was used so we could calculate the dynamic responses of the model under the time-based loadings.

RESULTS

Maximum Von Mises stress (MVMS) was observed in the cortical bone in the crest adjacent to the implant neck in both models. There was no

TABLE 3

Amount and area of maximum stress on bone while loading on crowns

Abutment type	One-piece	Multi-piece
Maximum Von Mises stress (MVMS)	23.3 MPa	27.9 MPa
Concentration area of MVMS	Distobuccal area CR-BCB*	Distobuccal area CR-BCB*

*CR-BCB: crest ridge-buccal cortical bone.

significant stress on the cancellous bone. The amount of maximum stress on the one-piece abutment was concentrated on a thin part of the distobuccal area of the cortical bone and was 23.3 MPa. In the multi-piece abutment model, the maximum stress was observed in the distobuccal area, which was 27.9 MPa. In the multi-piece abutment model, the stress distribution area was higher compared to the one-piece abutment model.

The amount and area of maximum stress on the bone while loading on the crowns are shown in Table 3.

Moreover, the pattern of distribution and amount of stress in the abutment models and bone structure surrounding the implant while loading on the crowns is shown in Figures 9 and 10.

DISCUSSION

The maximum stress on the cortical bone in the crest area under loading on crowns in the multi-piece abutment was higher than that of the one-piece abutment (27.9 MPa vs 23.3 MPa). This result is consistent with the Misch study,^{19,20} which states that cemented restorations decrease the strain on the crest area of implants, while the screw-retained restorations increase the exerted force on the implant and causes bone loss, implant loosening, and fracture of parts.

In Cehreli and colleagues' study,¹³ findings demonstrated that the two-piece implants experience higher mechanical stress than did one-piece implants under loading. They also revealed that the two-piece model exhibits wider distribution of stresses in the Morse taper region. In their study, the two-piece implant model experienced higher Von Mises stresses under loading.

The Hansson⁷ study showed that the peak bone stresses arise in the upper part of the crestal bone. However, in a finite element study of an axially loaded dental implant, Hansson²¹ found that the

location of the peak bone-implant interface shear stress depends on the design of the implant-abutment interface.

When compared to the Forest classification²² for cortical bone and their mechanical compatibility with the strain, the amount of maximum stress in our study was close to the osteogenesis stimulus threshold (24.8 MPa). Amounts of maximum stress on the bone surrounding the one-piece and multi-piece models are 1957 and 2461 microstrains, respectively. These values are in the range of mild loading of cortical bone (1500–3000 microstrains). In this range, the bone formation and remodeling inhibition were both observed; as a result, the strength and density of the bone might decrease. The bone in this area is woven, so remodeling will be done in a way that the bone can bear the strain.⁵

The implant fixture ITI is pure titanium (grade 4), which has an elastic coefficient 8.5 times more than the cortical bone. Based on a mechanical law, when two materials with different elastic coefficients are next to each other, there is no material between them, and one is under pressure, the stress in the contact zones will be increased.²³ This stress will form a U or V pattern, and maximum stress will be on the first point of contact. This was shown with photoelastic studies and three-dimensional finite element analysis.²³ Therefore, the high stress concentration on the cortical bone adjacent to the implant neck in the crest area can be justified by this theory.

These results are also consistent with several studies: Sevimay et al,¹ Koca et al,¹⁵ Kitamura et al,²⁴ and Judzbalys et al.²⁵ In these researchers' studies, maximum stress was observed in the cortical bone adjacent to the implant neck, and no significant stress was observed on the cancellous bone surrounding the implant. As the strength of the cortical bone is more than that of the cancellous bone, the cortical bone bears more tension than the cancellous bone. This is due to the elastic coefficient difference of these two types of bones.

CONCLUSION

According to the results of our study, the abutment type (the abutment-implant connection interface) influences the amount and distribution of stress on bone structures during dynamic loading.

The multi-piece abutment causes more stress concentration (27.9 MPa) and the one-piece solid abutment causes less stress concentration (23.3 MPa) on the cortical bone adjacent to the implant neck.

The type of abutment influences the stress distribution. In the multi-piece abutment model, a wider area of stress on the bone was observed compared to the one-piece abutment.

Effectiveness of using dynamic analysis was demonstrated. This analysis could predict the effect of the abutment type on the amount and distribution of stress on the bone during loading.

Further investigations are needed to address the optimal design of abutments using FEM method.

ABBREVIATIONS

FEM: finite element model

RN: regular neck

MVMS: maximum Von Mises stress

REFERENCES

- Sevimay M, Turhan F, Kilicarslan MA, Eskitascioglu G. Three-dimensional finite element analysis of the effect of different bone quality on stress distribution in an implant. Supported crown. *J Prosthet Dent*. 2005;93:227–234.
- Misch CE. Density of bone: effect on treatment plans, surgical approach, healing, and progressive bone loading. *Int J Oral Implantol*. 1990;6:23–31.
- Isidor F. Influence of force on peri-implant bone. *Clin Oral Imp1 Res*. 2006;17(suppl 2):8–18.
- Bozkage D, Muftu S. Mechanics of the tapered interference fit in dental implants. *J of Biomechanics*. 2003;36:1649–1658.
- Misch CE. Principles of cement-retained fixed implant prosthodontics. In: Misch CE, ed. *Dental Implant Prosthetics*. 3rd ed. Maryland Heights, Mo: Mosby; 2005:422.
- Chun HJ, Shin HS, Han CH, Lee SH. Influence of implant abutment type on stress distribution in bone under various loading conditions using finite element analysis. *Int J Oral Maxillofac Implants*. 2006;21:195–202.
- Hansson S. A conical implant-abutment interface at the level of the marginal bone improves the distribution of stresses in

the supporting bone. An axisymmetric finite element analysis. *Clin Oral Implants Res*. 2003;14:286–293.

8. Williams KR, Williams ADC. Implant response of a dental implant in bone by numerical analysis. *Biomaterial*. 1997;18:715–719.

9. Kitagawa T, Tanimoto Y, Odaki M, Nemoto K, Aida M. Influence of implant/abutment joint designs on abutment screw loosening in a dental implant system. *J Biomed Mater Res Part B: Appl Biomater*. 2005;758:457–403.

10. Huang HM, Lee SY, Yeh CY, Lin CT. Resonance frequency assessment of dental implant stability with various bone qualities: a numerical approach. *Clin Oral Implants Res*. 2002;13:65–74.

11. Quresma SE, Cury PR, Sendyk WR, Sendyk C. A finite element analysis of two different dental implants: stress distribution in the prosthesis, abutment, implant, and supporting bone. *J Oral Implantol*. 2008;34:1–6.

12. Alkan I, Sertgöz A, Ekici B. Influence of occlusal forces on stress distribution in preloaded dental implant screws. *J Prosthet Dent*. 2004;91:319–325.

13. Cehreli MC, Akça K, Iplikçioğlu H. Force transmission of one- and two-piece Morse-taper oral implants: a nonlinear finite element analysis. *Clin Oral Implants Res*. 2004;15:481–489.

14. Zienkiewicz OC, Taylor RL. *The Finite Element Method: Its Basis and Fundamentals*. Vol 1. 5th ed. Oxford, UK: Butterworth-Heinemann; 2000.

15. Koca OL, Eskitascioglu G, Usumez A. Three-dimensional finite-element analysis of functional stresses in different bone locations produced by implant placed in the maxillary posterior region of the sinus floor. *J Prosthet Dent*. 2005;93:38–44.

16. Lekholm U, Zarb GA. Patient selection and preparation. In: Branmark PI, Zarb GA, Albreksson T, eds. *Tissue-Integrated Prostheses: Osseointegration in Clinical Dentistry*. Hanover Park, Ill: Quintessence; 1985:199–209.

17. Ash MM. *Wheeler's Dental Anatomy; Physiology and Occlusion*. 6th ed. Philadelphia, Pa: WB Saunders;1984.

18. Misch CE. Available bone and implant dentistry. In Misch CE, ed. *Dental Implant Prosthetics*. 3rd ed. Maryland Heights, Mo: Mosby; 2005:107–109.

19. Misch CE. Principles of cement-retained fixed implant prosthodontics. In: Misch CE, ed. *Dental Implant Prosthetics*. 3rd ed. Maryland Heights, Mo: Mosby; 2005:419.

20. Misch CE. Principles for screw-retained prostheses. In: Misch CE, ed. *Dental Implant Prosthetics*. 3rd ed. Maryland Heights, Mo: Mosby; 2005:460.

21. Hansson S. Implant-abutment interface: biomechanical study of flat top versus conical. *Clin Implant Dent Relat Res*. 2000; 2:33–41.

22. Misch CE, Bidez MW. A scientific rationale for dental implant design. In: Misch CE, ed. *Dental Implant Prosthetics*. 3rd ed. Maryland Heights, Mo: Mosby; 2005:338–340.

23. Misch CE. Stress factors: influence on treatment planning. In: Misch CE, ed. *Dental Implant Prosthetics*. 3rd ed. Maryland Heights, Mo: Mosby; 2005:76–77.

24. Kitamura E, Stegaroiu R, Nomura S, Miyakawa O. Biomechanical aspects of marginal bone resorption around osseointegrated implants: three-dimensional finite element analysis. *Clin Oral Implants Res*. 2004;15:401–412.

25. Gintaras J, Ricardas K, Valdas E, Aune R. Stress distribution in bone: single-unit implant prostheses veneered with porcelain or a new composite material. *Implant Dent*. 2005;14:166–175.