Finite Element Stress Analysis according to Apical-coronal Implant Position

Tae-Ho Kang, Su-Gwan Kim

Department of Oral and Maxillofacial Surgery, Oral Biology Research Institute, College of Dentistry, Chosun University

Abstract

Purpose: The purpose of this study was to evaluate the influence of apical-coronal implant position on the stress distribution after occlusal and oblique loading.

Materials and Methods: The cortical and cancellous bone was assumed to be isotropic, homogeneous, and linearly elastic. The implant was apposed to cortical bone in the crestal region and to cancellous bone for the remainder of the implant-bone interface. The cancellous core was surrounded by 2-mm-thick cortical bone. An axial load of 200 N was assumed and a 200-N oblique load was applied at a buccal inclination of 30 degrees to the center of the pontic and buccal cusps. The 3-D geometry modeled in Iron CAD was interfaced with ANSYS.

Results: When only the stress in the bone was compared, the minimal principal stress at load Points A and B, with a axial load applied at 90 degrees or an oblique load applied at 30 degrees, for model 5. The von Mises stress in the screw of model 5 was minimal at Points A and B, for 90- and 30-degree loads. When the von Mises stress of the abutment screw was compared at Points A and B, and a 30-degree oblique load, the maximum principal stress was seen with model 2, while the minimum principal stress was with model 5. In the case of implant, the model that received maximum von Mises stress was model 1 with the load Point A and Point B, axial load applied in 90-degree, and oblique load applied in 30-degree.

Discussion and Conclusions: These results suggests that implantation should be done at the supracrestal level only when necessary, since it results in higher stress than when implantation is done at or below the alveolar bone level. Within the limited this study, we recommend the use of supracrestal apical-coronal positioning in the case of clinical indications.

Key words

Finite element stress analysis, Apical-coronal implant position

INTRODUCTION

The major contributing factor leading to late failure of implant-supported restorations seems to be a lack of understanding of biomechanical concepts[1]. Bioengineering studies have been useful for determining the optimal prosthesis design and understanding the response of biologic tissues to relevant forces[2-4]. Biomechanical factors play an important role in the long-term survival of dental implants. The selection of implant position is critical for the longevity and stability of an implanted prosthesis[5].

* Corresponding author

Su-Gwan Kim D.D.S., Ph.D
*Dept. of OMFS, College of Dentistry, Chosun University,
421, SeoSeogDong, DongGu, GwangJu City, 501-825, Korea
Tel: 82-62-220-3815   FAX: 82-62-224-9172
E-mail: SGCKIM@mail.chosun.ac.kr*

Two different kinds of implant placement methods are used in the field of recent oral implantology: submerged (two-stage surgical procedure) and non-submerged (one-stage surgery) type implants[6]. Multicenter studies have reported satisfactory success rates for both protocols and a similar loss of crestal peri-implant bone after occlusal loading (0.5 to 1.5 mm). The major problem with the submerged approach is the countersinking process needed to submerge the implant. In order to accommodate the implant head and cover screw, the crestal bone must be widened approximately. When the labio-lingual width of the alveolar bone is less than 6 mm, the bone is too thin for implantation of standard diametered fixtures and the bone around the implant is eventually absorbed. Consequently, the bone loss would be less than 0.5 mm. Davarpanah and coworkers[7] first reported the concept of supracrestal placement in detail; and clinical experience shows that bone loss is clearly reduced with implanta-

tion using supracrestal placement.

Even when the labio-lingual width of alveolar bone is narrow, countersinking is unnecessary with supracrestal placement, and the biological width is maintained, reducing bone loss.

Recently, supracrestal apical-coronal positioning of the implant collar has been proposed for posterior sectors using submerged implants[7]. The advantages of supracrestal apical-coronal positioning are as follows: little or no countersinking is needed; there is no need for underseating when connecting the abutment; soft tissue management is simple; the surgical procedure and suturing are simple; longer implants can be used; the clinical crown-implant ratio and implant anchorage surface increase; the second surgical phase and prosthetic restoration are facilitated; the amount of peri-implant crestal bone loss is reduced or limited; and there is optimal initial stability[7].

Three-dimensional finite element stress analysis (FEA) is a technique that is used to solve engineering problems. It can be used in numeric simulations to determine stress and displacement, via its ability to model geometrically complex structures. The method has been extensively used to study the biomechanics of dental implants and has many advantages over other methods[8].

A few FEA studies have built a 3-D finite element model database of the mandible[9-11]. However, as far as I know, no FEA analysis has tried to study the stress distribution according to apical-coronal implant position.

The purpose of this study was to evaluate the influence of apical-coronal implant position on the stress distribution after occlusal and oblique loading.

MATERIALS AND METHODS

Three-dimensional (3-D) geometry formation

A mandible taken from a fresh cadaver was digitized using a surface scanner. Modeling was done using 3-D computer-assisted design (IronCAD ver. 6.0, Atlanta, GA). The model consisted of a 30-mm piece of bone, which included 15 mm on each side of the tooth center. In addition, the mandibular bone model was divided into cortical and cancellous bone for a more detailed analysis. Computed tomography (CT) images of actual mandibular bone were used to make more accurate model. The five lines in Fig. 1 were designated as lines 1 to 5 from top to bottom, respectively. The 3-D geometry of the full body for the external implant/abutment connection is shown in Fig. 2.

Construction of 3-D finite element analysis model

The cortical and cancellous bone was assumed to be isotropic, homogeneous, and linearly elastic. The implant was apposed to cortical bone in the crestal region and to cancellous bone for the remainder of the implant-bone interface. The cancellous core was surrounded by 2-mm-thick cortical bone. The cancellous bone was classified as dense because of the anatomical structure of the mandible.
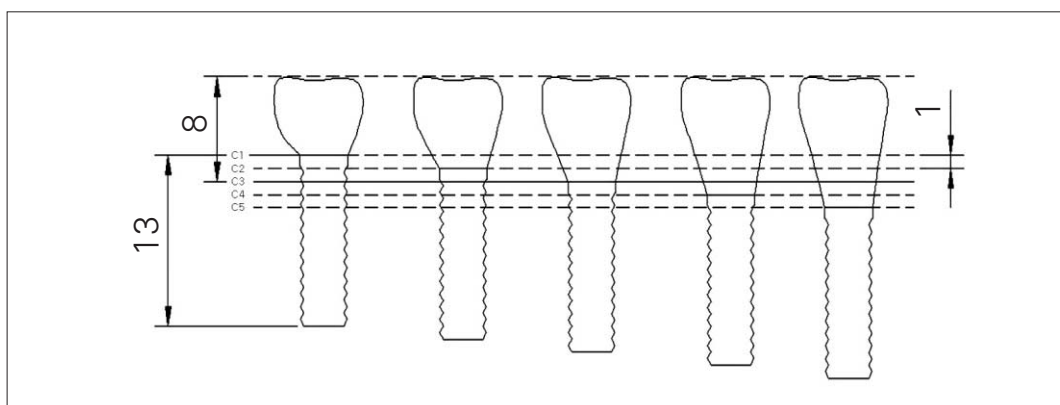

Fig. 1. Apical-coronal implant position. C1 = 2 mm supracrestal implant position; C2 = 1 mm supracrestal implant position; C3 = crest of cortical bone; C4 = 1 mm subcrestal implant position; C5 = 2 mm subcrestal implant position.

To simulate complete osseointegration, the implants were rigidly anchored along the entire interface in the bone model (Fig. 3). Young's modulus for cortical and cancellous bone was assumed to be 13,700 and 1,370 MPa, respectively. Poisson's ratio was 0.3 for both (Table 1).

The 3-D geometry modeled in Iron CAD was interfaced with ANSYS ver. 7.0 (ANSYS Inc., Canonsburg, PA, USA) to generate a grid. This process considered consistent, so that the number of elements and nodes would be similar for each model. Consequently, the element used was a tetrahedral element with eight nodes (Table 2).

Implants

AVANA dental implants (Osstem, Busan, Korea), external fixture, were designed at five different bone depths for each implant/abutment connection. Self-tapping, screw-type implants 11.5 mm long and 4 mm in diameter was selected as an external type fixture (Fig. 3).

Loads and boundary conditions (Fig. 4)

An axial load of 200 N was assumed (Fig. 5); this is referred to as the 90-degree axial load. A 200-N oblique load was applied at a buccal inclination of 30 degrees to the center of the pontic and buccal cusps (Fig. 6). Loads were applied at Points A and B. Both ends of the bone were bounded (x-, y-, z-dimensions).

Solution

Solutions were obtained using ANSYS. Both overall and each component (bone, screw, abutment screw, implant) stress were analyzed and expressed in von
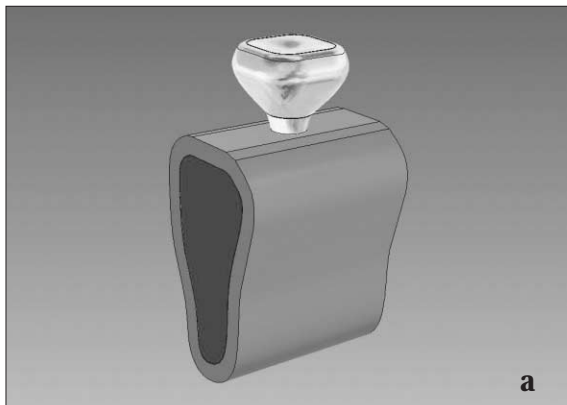

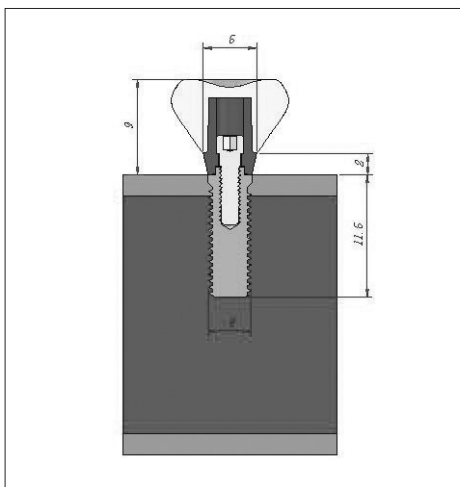**Fig. 2.** 3-D geometry of full body (a) and 3-D finite element model (b).

Fig. 3. Surface dimensions of the external fixture.

Table 1. Material Properties

	Young's Modulus (MPa)	Possion's Ratio
Compact bone	13,700	0.3
Cancellous bone	1,370	0.3
Implant	115,000	0.35
Titanium screw	115,000	0.35
Gold crown	96,600	0.35

Fig. 4. Points subjected to loading.

Table 2. Numbers of Elements and Nodes in the Mandible Model

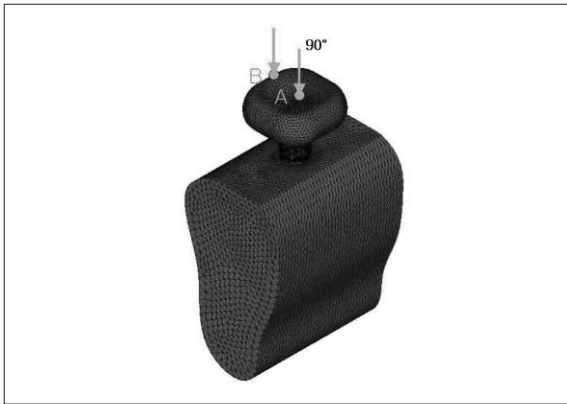|	Node	Element
Model 1	53,816	282,908
Model 2	53,845	287,289
Model 3	54,066	290,604
Model 4	54,394	292,513
Model 5	53,537	287,786

Fig. 5. Load when a 90-degree axial force was applied.
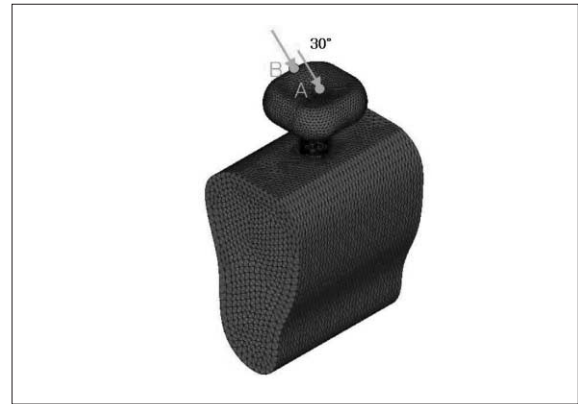
Fig. 6. Load when a 30-degree oblique force was applied.

Mises stress (equivalent stress). The stress patterns were displayed as contour lines with different colors connecting regions of equal stress within defined limits.

RESULTS

To evaluate the stress distribution, the magnitude of the stress concentration was presented as the minimum (compressive stress) and maximum (tensile stress) principal stress.

Both overall and each component (bone, screw, abutment screw, implant) stress were expressed from Fig. 7 to Fig. 11. When only the stress in the bone was compared, the minimal principal stress at load Points A and B, with a axial load applied at 90 degrees or an oblique load applied at 30 degrees, for model 5 (Figs 12 and 13).

The von Mises stress in the screw of model 5 was minimal at Points A and B, for 90- and 30-degree loads (Figs 14 and 15). When the von Mises stress of the abutment screw was compared at Points A and B, and a 30-degree oblique load, the maximum principal stress was seen with model 2, while the minimum principal stress was with model 5 (Figs 16 and 17). In the case of implant, the model that received maximum von Mises stress was model 1 with the load Point A and Point B, axial load applied in 90-degree, and oblique load applied in 30-degree (Fig. 18).

Overall, the maximum von Mises stress in the models increased in the order of 5<4<3<2<1. Moreover, for an applied load at the same point, the stress was greater with an oblique load than with a axial load.

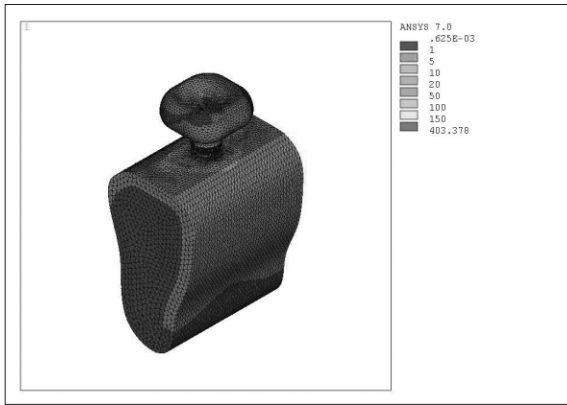**Fig. 7.** Equivalent stress in overall form (model 1, 30-degree, Point A).
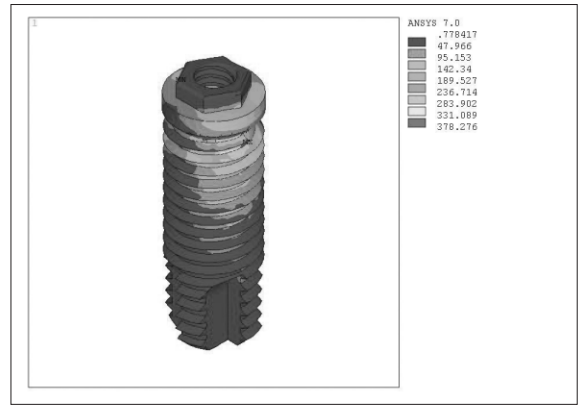

Fig. 8. Equivalent stress in implant (model 1, 30-degree, Point A).
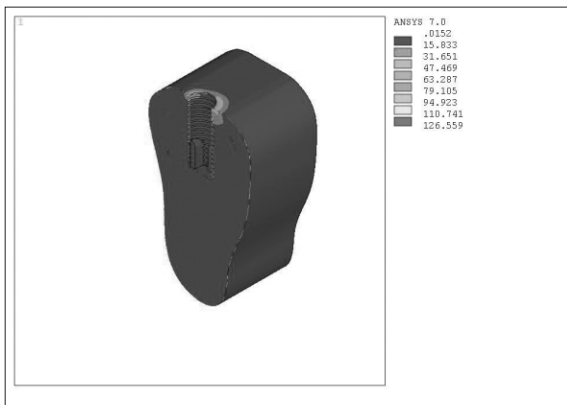

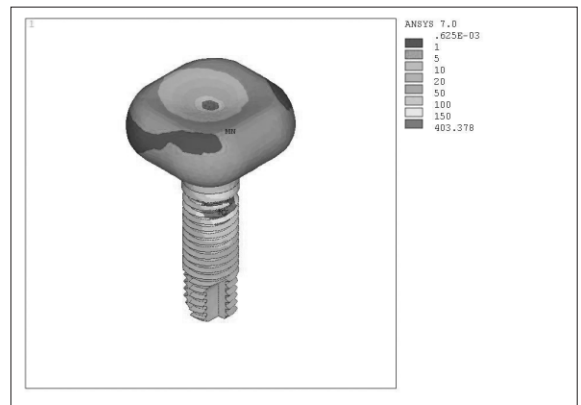**Fig. 9.** Equivalent stress on bone surface (model 1, 30-degree, Point A).

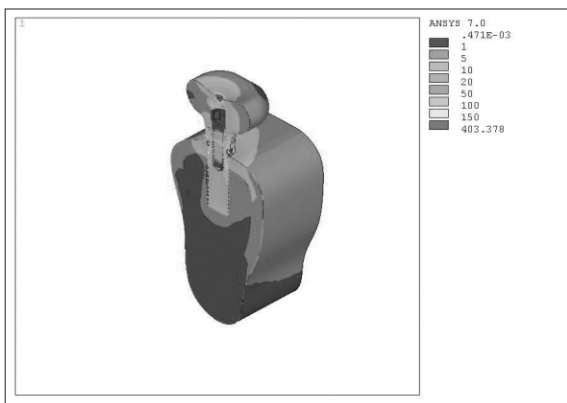**Fig. 10.** Each component (implant, abutment, and crown) stress.

Fig. 11. Equivalent stress on overall surface (model 1, 30-degree, Point A).
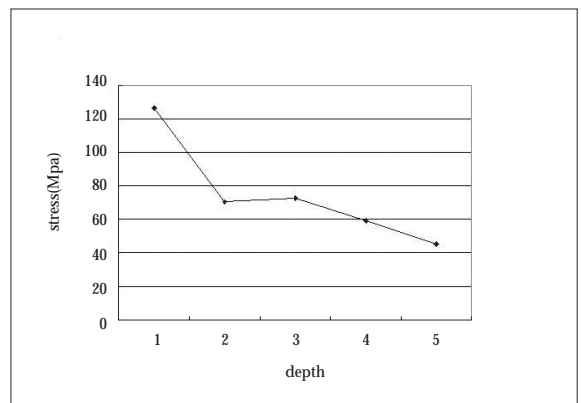

Fig. 12. Comparison of the von Mises stress according to apical-coronal implant position (loading site=A, load=30-degree).
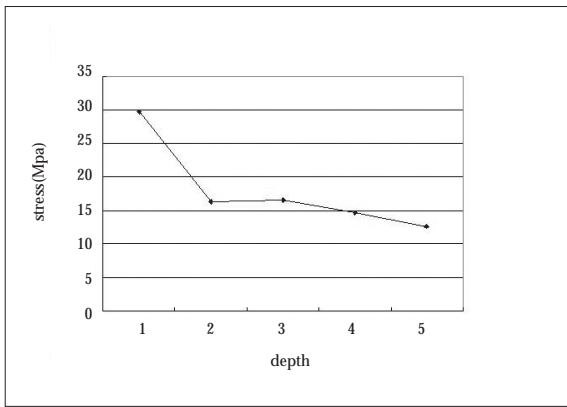
Fig. 13. Comparison of the stress according to apical-coronal implant position (loading site=A, load=90-degree).
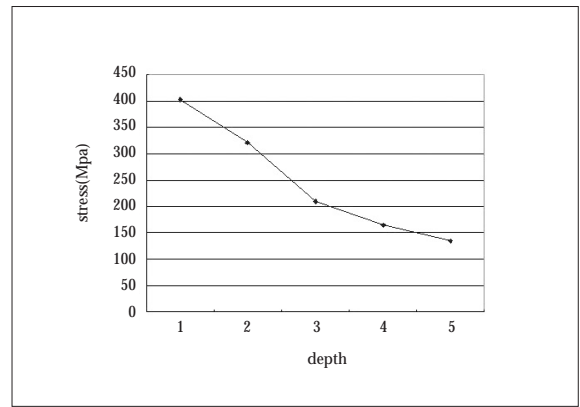

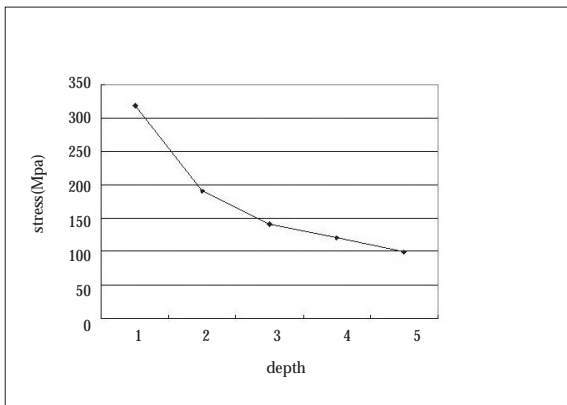**Fig. 14.** Stress comparison for an external screw (loading site=A, load=30-degree).

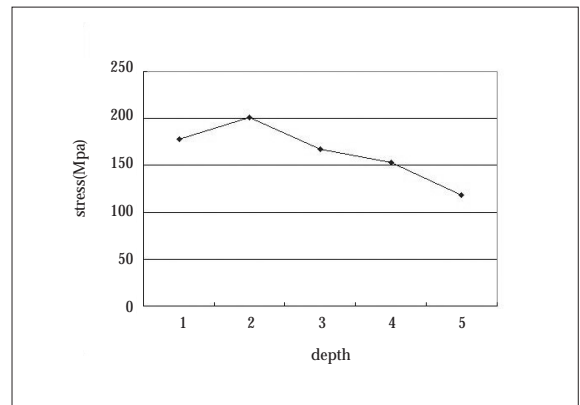**Fig. 15.** Stress comparison for an external screw (loading site=B, load=90-degree).

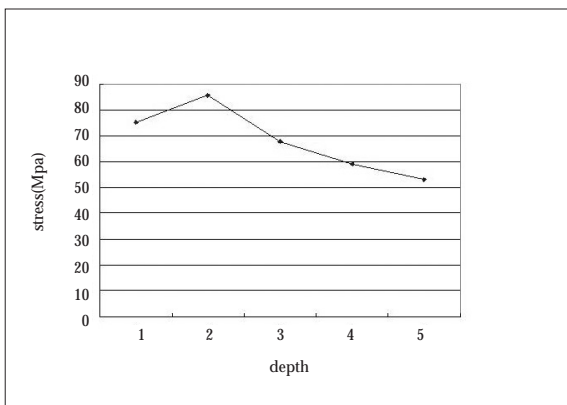**Fig. 16.** Stress differences in the abutment screw (loading site=A, load=30-degree).

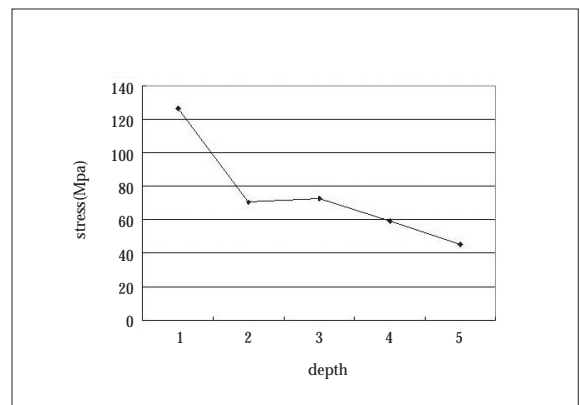**Fig. 17.** Stress differences in the abutment screw (loading site=B, load=30-degree).

Fig. 18. Von Mises stress in the implant (loading site=A, load=30-degree).

DISCUSSION

Finite element analysis (FEA) is a precise method for evaluating dental implant systems. It has been used extensively to predict the biomechanical performance of various dental implant designs and the effect of clinical factors on implant success[12,13]. 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 those results to clinical situations[13].

However, FEA has inherent limitations. The stress values that actually cause biologic changes in bone, such as resorption and remodeling, are not known. In addition, the stress values calculated using FEA are not necessarily identical to actual values[14].

The von Mises stress is defined as the stress at which the deformation of ductile materials, such as metallic implants, begins. Since failure occurs when the von Mises stress exceeds the yield strength of an implant, von Mises stress criteria are important for interpreting the stresses that occur within the implant[1]. The principal stress can be used to distinguish tensile and compressive stress.

In order to improve osseointegration, recent studies have focused on implant position, shape, and surface characteristics[2-4,15]. Stress around implants may lead to bone resorption and implant loss[16]. Therefore, determining the stress distribution and intensity is important for understanding the process that leads to the loss of integration.

Insufficient bone volume and increased prosthetic space may be encountered in the placement of supracrestal implants (3 to 4 mm); consequently, the process can be combined with a guided tissue regeneration procedure in order to increase the alveolar ridge height[17].

Supracrestal apical-coronal positioning of the collar improves the clinical crown-implant ratio and the implant anchorage surface. In posterior sectors, an implant that is 1.5 mm longer may be used, and this markedly increases anchorage. The combination of a rough surface and a supracrestal apical-coronal position increases implant anchorage considerably[7].

The disadvantages of supracrestal apical-coronal positioning are: osseointegration could fail due to overload; failure could result when primary stability is insufficient;

there is the possibility of metal exposure in the esthetic zone (anterior area); and implant micromovement can occur[7].

Contraindications for this surgical technique are: when there is a minimum difference in the crestal bone level between the implant and adjacent teeth; when the intermaxillary distance is less than 6 mm; or when the mucosa is very thin[7]. Supracrestal apical-coronal positioning should be used selectively when bone quality is good, primary stability is sufficient, and the intermaxillary distance is sufficient.

Mericske-Stern and coworkers[18] explored the occlusal force in a group of partially edentulous patients restored using ITI implants supporting a fixed partial prosthesis. They found that the range of occlusal force on the second premolar and molar teeth was 210-400 N and 130-395 N, respectively. In this study, a static load of 200 N was applied at a buccal inclination of 30 degrees and 90 degree to the buccal cusps.

Since the number of clinical cases needing supracrestal apical-coronal positioning is increasing, this study examined the utility of FEA. In this study, the maximum von Mises stress occurred with model 1, especially at a 30-degree oblique load. Moreover, the minimum von Mises stress occurred with model 5. For an abutment screw, the maximum principal stress occurred with model 2, while the minimum principal stress was with model 5.

Overall, the maximum principal stress increased as the model decreased from 5 to 1 and the minimum principal stress decreased as the model increased from 1 to 5.

Our finite element analysis suggests that implantation should be done at the supracrestal level only when necessary, since it results in higher stress than when implantation is done at or below the alveolar bone level. Within the limited this study, we recommend the use of supracrestal apical-coronal positioning in the case of clinical indications. Nevertheless, an implant that is positioned too deep is not good for the gingiva in the long term because of difficulties in tissue maintenance and tissue management. Crestal bone loss is the most common cause of implant failure when osseointegration has been achieved[19]. Therefore, further animal and clinical studies are needed. Our analysis considered implantations of the same length; additional analysis is needed for cases in which the length of implant increases as the clinical crown becomes shorter and vice versa.

REFERENCES

1. Akca K, Iplikcioglu H: Finite element stress analysis of the influence of staggered versus straight placement of dental implants. Int J Oral Maxillofac Implants 2001;16:722-730.
2. Cooper LF, Masuda T, Yliheikkila PK, Felton DA: Generalizations regarding the process and phenomenon of osseointegration. Part II. *In vitro* studies. Int J Oral Maxillofac Implants 1998;13:163-174.
3. Masuda T, Yliheikkila PK, Felton DA, Cooper LF: Generalizations regarding the process and phenomenon of osseointegration. Part I. *In vivo* studies. Int J Oral Maxillofac Implants 1998;13:17-29.
4. 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.
5. Ciftci Y, Canay S: The effect of veneering materials on stress distribution in implant-supported fixed prosthetic restorations. Int J Oral Maxillofac Implants 2000;15:571-582.
6. Boioli LT, Penaud J, Miller N: A meta-analytic, quantitative assessment of osseointegration establishment and evolution of submerged and non-submerged endosseous titanium oral implants. Clin Oral Implants Res 2001;12:579-588.
7. Davarpanah M, Martinez H, Tecucianu JF: Apical-coronal implant position: recent surgical proposals. Technical note. Int J Oral Maxillofac Implants 2000;15:865-872.
8. Williams KR, Watson CJ, Murphy WM, Scott J, Gregory M, Sinobad D: Finite element analysis of fixed prostheses attached to osseointegrated implants. Quintessence Int 1990;21:563-570.
9. Kregzde M: A method of selecting the best implant prosthesis design option using three-dimensional finite element analysis. Int J Oral Maxillofac Implants 1993;8:662-673.
10. Meijer HJ, Starmans FJ, Steen WH, Bosman F: A three-dimensional, finite-element analysis of bone around dental implants in an edentulous human mandible. Arch Oral Biol 1993;38:491-496.
11. van Zyl PP, Grundling NL, Jooste CH, Terblanche E: Three-dimensional finite element model of a human mandible incorporating six osseointegrated implants for stress analysis of mandibular cantilever prostheses. Int J Oral Maxillofac Implants 1995;10:51-57.
12. Holmgren EP, Seckinger RJ, Kilgren LM, 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. J Oral Implantol 1998; 24:80-88.
13. 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.
14. Akca K, Iplikcioglu H: Evaluation of the effect of the residual bone angulation on implant-supported fixed prosthesis in mandibular posterior edentulism. Part II: 3-D finite element stress analysis. Implant Dent 2001;10:238-245.
15. De Leonardis D, Garg AK, Pecora GE: Osseointegration of rough acid-etched titanium implants: 5-year follow-up of 100 minimatic implants. Int J Oral Maxillofac Implants 1999;14:384-391.
16. Lai H, Zhang F, Zhang B, Yang C, Xue M: Influence of percentage of osseointegration on stress distribution around dental implants. Chin J Dent Res 1998;1:7-11.
17. Martinez H, Davarpanah M, Missika. Implant treatment for the posterior regions: classification. In: Davarpanah M, Martinez H, Kebir M, Tecucianu JF (eds). Clinical Manual of Implant Dentistry. Chicago: Quintessence, 2003:156.
18. Mericske-Stern R, Assal P, Mericske E, Burgin W: Occlusal force and oral tactile sensibility measured in partially edentulous patients with ITI implants. Int J Oral Maxillofac Implants 1995;10:345-353.
19. O' Mahony AM, Williams JL, Spencer P: Anisotropic elasticity of cortical and cancellous bone in the posterior mandible increases peri-implant stress and strain under oblique loading. Clin Oral Implants Res 2001;12:648-657.