Finite element stress analysis of internal implant/abutment connection according to different apical-coronal implant position

Su-Gwan Kim, Dong-Cook Choi, Byung-Ock Kim*, Sang-Ho Oh**

Dept. of Oral & Maxillofacial Surgery, Rehabilitation of Tissue Defects in Oral and Maxillofacial Region, BK21,

Department of Periodontology*, Department of Prosthodontics** School of Dentistry, Chosun University

Ⅰ. INTRODUCTION

The supracrestal apical-coronal positioning of the implant collar improves the clinical crown/implant ratio and the implant anchorage surface. The combination of a rough surface and a supracrestal apical-coronal position considerably increases the implant anchorage[1]. The disadvantages of supracrestal apical-coronal positioning are that osseointegration can fail owing to overload; failure can result owing to insufficient primary stability; metal may become exposed in the aesthetic zone (anterior area); and implant micromovements can occur[1].

The fixture and abutment can be connected externally or internally. From a design perspective, an internal connection is slightly advantageous in one-stage surgery. Furthermore, the risk of loosening is reduced, because an internal connection results in a stronger connection than does an external system and less plaque is deposited owing to fewer microgaps[2].

The external hexagon of the Brånemark implant pattern is still the standard method of connection. The connection of the abutment closely contacts the inner wall of the implant, forming a strong, stable surface that is resistant to micromovements; the loosening of the connecting part in the deep internal joint occurs with low frequency; and the connecting part resists bending forces. In cement-retained prostheses, the stability of the connecting part of the abutment is particularly important, and thus the use of an internal joint is recommended. Particularly, with conical seal type or ring type sealing, such as with Frialit implants, the abutment-implant interface is sealed tightly, decreasing the entry of microorganisms. It is easier to connect the superstructure of the fixture with an internal joint that with an external one, and, in the case of a cone screw, X-rays are not required to confirm an accurate attachment[3].

The finite element analysis (FEA) method has proved to be a useful tool in estimating stress levels around implants[4-8]. It involves the development of a mathematical model of a continuous structure divided into a system of discrete components or elements. These components are connected at nodal points, where stresses and displacements are determined. The accuracy of the 3-dimensional method is a proportional to the number of nodes and elements in the mathematical model[9].

Several FEA studies have built a 3-D finite element model database of the mandible[10-12]. To the best of author's knowledge, however, no FEA analysis has examined the stress distribution according to the apical-coronal implant position using an implant with an internal connection.

This study evaluated the influence of the apical-coronal implant position on the stress distribution after vertical and oblique loading, using an implant with an internal connection.

Ⅱ. MATERIALS AND METHODS

Three-dimensional (3-D) geometry formation

A mandible taken from a fresh cadaver was digitized using a surface scanner. Modeling was performed 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. The mandibular bone model was divided into cortical and cancellous bone for a more detailed analysis. Computed tomography (CT) images of the actual mandibular bone were used to make a more accurate model. The five lines (C1, C2, C3, C4, C5) in Fig. 1 were designated as lines 1 to 5 (1 mm distance) from top to bottom, respectively. The 3-D geometry of the full body of the internal fixture is shown in Fig. 2.

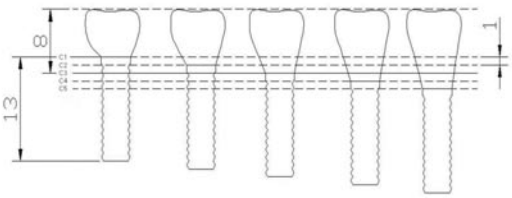
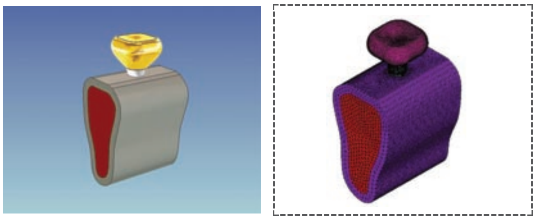


Fig. 1. Apical–coronal implant position: implant position C1 = 2 mm supracrestal, C2 = 2 mm supracrestal, C3 = crest of cortical bone, C4 = 1 mm subcrestal, C5 = 2 mm subcrestal.



a b

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

Construction of the 3-D FEA 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 based on the anatomical structure of the mandible.

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

The 3-D geometry modeled with IronCAD was interfaced with ANSYS ver. 7.0 (ANSYS Inc., Canonsburg, PA, USA) to generate a grid. In this process, the numbers of elements and nodes in each model were similar. Consequently, a tetrahedral element with eight nodes was used (Table II).

Table I. 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
Resin	9,700	0.35

Table II. Numbers of nodes and elements in the mandible model

	Node	Element
Model 1	41,894	213,281
Model 2	43,060	221,151
Model 3	44,228	229,059
Model 4	44,527	232,023
Model 5	45,310	237,653

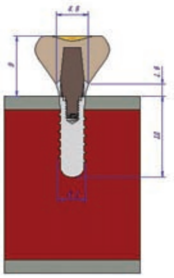


Fig. 3. Surface dimensions of the internal fixture.

Implants

AVANA dental implants (Osstem, Busan, Korea)with an internal fixture were designed for five different bone depths for each implant/abutment connection. Self-tapping, screw-type implants measuring 12 mm long and 4.1 mm in diameter were selected as the internal-type fixture (Fig. 3).

Loads and boundary conditions (Fig. 4)

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

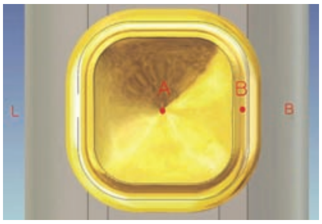


Fig. 4. Points subjected to loading. Point A: the center of the pontic, Point B: buccal cusp

Fig. 5. Load owing to a 90° vertical force.
Point A: the center of the pontic, Point B: buccal cusp

Fig. 6. Load owing to a 30° oblique force.
Point A: the center of the pontic, Point B: buccal cusp

Solution

Solutions were obtained using ANSYS. Both the overall stress and that on each component (bone, abutment screw, and implant) were analyzed and expressed as von Mises stress (equivalent stress). The stress patterns were displayed as contour lines with different colors connecting regions of equal stress within defined limits.

Ⅲ. RESULT

To evaluate the stress distribution, the magnitude of the stress concentration was presented as the minimum (compressive stress) and maximum (tensile stress) principal stresses. Both the overall stress and the stress on each component (bone, abutment screw, and implant) are shown in Figs. 7 to 11.

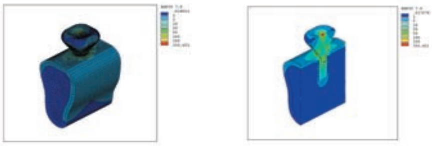


Fig. 7. Equivalent overall stress (Model 4, 30°, Point A).

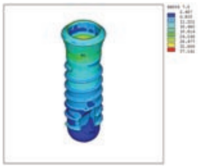


Fig. 8. Equivalent stress on the implant (Model 5, 90°, Point A).

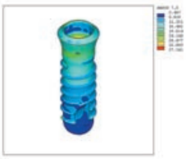


Fig. 9. Equivalent stress on the bone surface (Model 5, 90°, Point A).

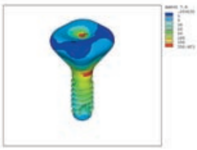


Fig. 10. Stress on each component (implant,abutment, and crown) (Model 4, 30°, Point A).

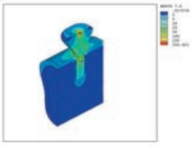


Fig. 11. Equivalent overall stress (Model 4, 30°, Point A).

1. When only the stress in the bone was compared, the minimal principal stress at load Point A and B, with an vertical load applied at 90° or an oblique load applied at 30°, occurred for the five internal implant/abutment connection models (Figs. 12 and 13).

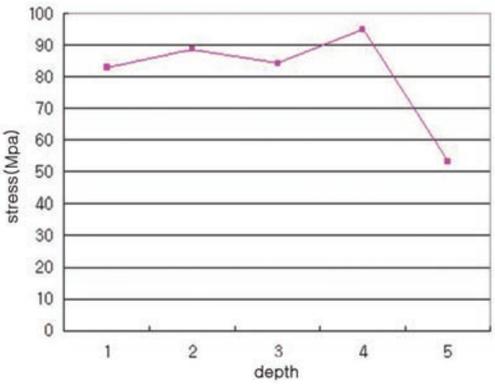


Fig. 12. Comparison of the von Mises stress according to the apical-coronal implant position in the bone(loading site = A, load = 30°).

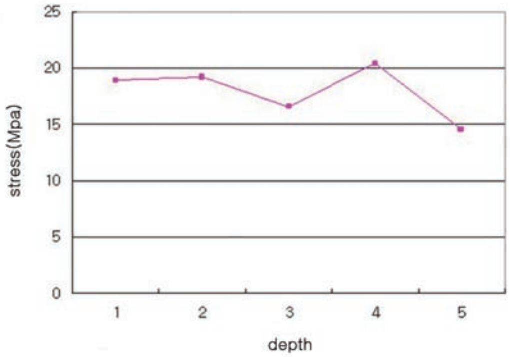


Fig. 13. Comparison of the stress according to the apical-coronal implant position in the bone (loading site = A, load = 90°).

2. Internal implant/abutment connection model 5 had the minimum principal stress with the loads at Points A and B, an vertical load applied at 90°, and an oblique load applied at 30° (Figs. 14 and 15). .

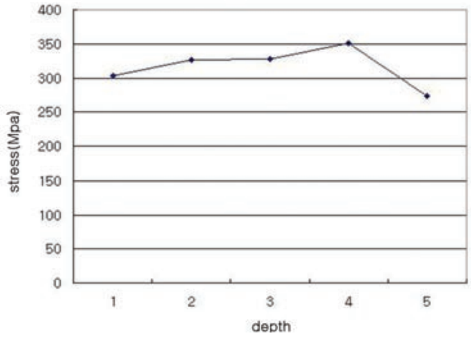


Fig. 14. Stress comparison for an internal screw (loading site = A, load = 30°).

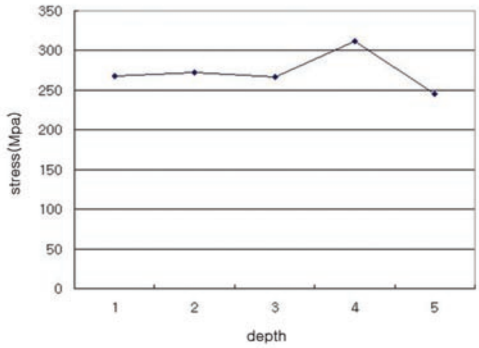


Fig. 15. Stress comparison for an internal screw (loading site = A, load = 90°).

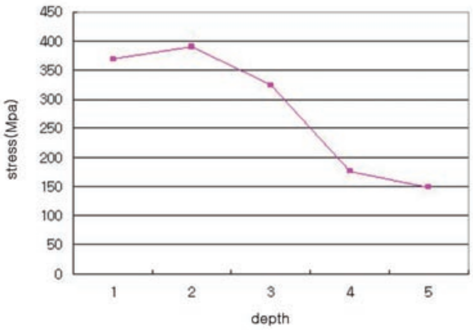


Fig. 16. Stress differences in the abutment screw (loading site = A, load = 30°).

3. When the von Mises stress of the abutment screw was compared at Points A and B, with a 30° oblique load, the maximum principal stress was seen with model 2, while the minimum principal stress was seen with model 5 (Figs. 16 and 17).

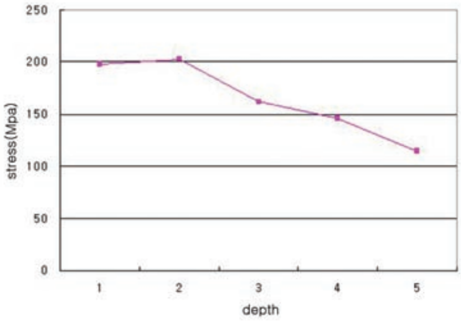


Fig. 17. Stress differences in the abutment screw (loading site = B, load = 30°).

4. For the implant, the model that received the minimum principal stress was model 5 with the load at Points A and B, an vertical load applied at 90°, and an oblique load applied at 30°(Fig. 18).

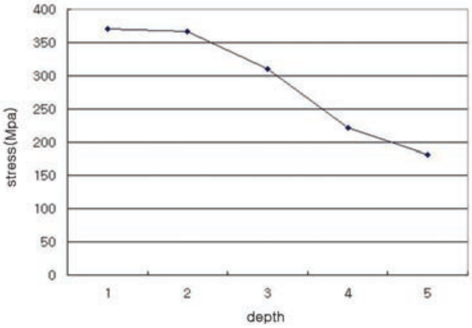


Fig 18. Von Mises stress in the implant (loading site = B, load = 90°).

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

Ⅳ. DISCUSSION

The significance of biomechanical aspects on the long-term success of osseointegrated implants has been emphasized. Three-dimensional FEA studies have contributed to aspects of implant biomechanics, through the use of unrealistic 100% bone contact FEA models with different element geometry, model sizes, material properties, displacement boundary conditions and loadings[13].

In order to improve osseointegration, recent studies have focused on implant position, shape, and surface characteristics[14-16]. Stress around implants may lead to bone resorption and implant loss[17].

Therefore, determining the stress distribution and intensity is important for understanding the process tha t leads to the loss of integration[2].

One of the keys to the long-term success of an implant is the choice of the system with respect to the superstructure and connection structure. Currently, internal type implants are popular, although this does not imply that internal implants are superior to external implants in all respects. Regardless of the choice of system, understanding the advantages and shortcomings of each system will facilitate the use of the most appropriate implant at the right time and place [2].

The biggest advantage of the internal connection design is that the screws loosen infrequently. In addition, the internal connection design reduces the vertical height of the platform for the restoration components, distributes the lateral force deeply into the implant, and protects the abutment screws by buffering vibrations applied to the abutment through an engagement with the implant wall. In addition, the stability and strength of the connection under conditions of bending are superior, and microbial entry at the implant-abutment interface is reduced[2]. When connecting this internal connection to the abutment, it is easier to sense when the attachment is accurate than with the external hex; however, as the connection is made within the fixture, the long-term assessment of any adverse effects on the strength of the fixture remains to be examined[3]. Following the development of the cone screw as an alternative connection method to the external hex, a variety of implant screws were designed. The most noteworthy are the cone screw, internal octagon, internal hexagon, cam cylinder, morse taper, spline, and resilient connections. Of these, the internal octagonal connection (Omniloc, Calciteck) and resilient connection (IMZ) are no longer used. As the wall of the octagonal design is thin, merely 0.6 mm, its diameter is small, and its profile is almost circular, this design was unable to properly resist rotary and lateral forces. In theory, with the IMZ resilient connection, the polyoxymethylene insert should neutralize the force applied to the implant by mimicking the periodontal ligament; unfortunately, this connection developed chronic maintenance problems and was redesigned with a metal insert[3].

Although the internal connection patterns vary widely, they can be classified grossly into two types: friction fit and passive fit. The friction fit types are the 8 morse taper (ITI, Ankylos, and Astra) and the Bicon with a rounded channel. A sufficient fit with tapers such as the rounded channel can be obtained with friction by inserting with pounding, but must be used for additional security with the morse taper pattern. The shortcoming of the friction-fit implants is that repeated relocation of the abutment is not feasible. To overcome this shortcoming, systems are available with an additional hexagonal or octagonal structure in addition to the morse taper (ITI, SwissPlus, Implantium, etc.)[2].

The passive-fit implant has a triangular, hexagonal, or octagonal antirotation structure, and its advantage is that the abutment in the passive state can be relocated within the implant. Examples of this type are CamLog (Trichannel), Replace Select (Trichannel), and Paragon (Hex, Octa)[2].

Conventional surgical technique is contraindicated when the intermaxillary distance is less than 6 mm or when the mucosa is very thin[1]. Supracrestal apical-coronal positioning should be used selectively when the bone quality is good, primary stability is sufficient, and the intermaxillary distance is sufficient. As the number of clinical cases requiring supracrestal apical-coronal positioning is increasing, we examined the utility of FEA in such cases. In this study, the maximum von Mises stress occurred with the internal implant/abutment connection in model 1, especially at a 30° oblique load. Moreover, the minimum von Mises stress occurred in model 5. For an abutment screw, the maximum principal stress occurred in model 2, while the minimum principal stress occurred in model 5. Overall, the maximum principal stress increased as the model number decreased from 5 to 1, and the minimum principal stress decreased as the model number increased from 1 to 5.

Within the limits of this study, implantation should be performed at the supracrestal level only when necessary, as it results in higher stress than does implantation at or below the alveolar bone level.

Nevertheless, an implant that is positioned too deeply is not good for the gingiva in the long term owing to difficulties in tissue maintenance and tissue management[18]. Thus, it is an alternative treatment that can be used for the implant length increases as the clinical crown becomes shorter.

REFERENCES

1. Davarpanah M, Martinez H, Tecucianu JF : Apical−coronal implant position: recent surgical proposals. Technical note. Int J Oral Maxillofac Implants 15:865−872, 2000.

2. 구강악안면 임프란트학. 대한나래출판사, 2004.

3. Kim YK, Hwang JW : Controversy in dental implant. Koonja publishing Co., p347, 2004.

4. Kim SG, Chon CG, Hwang GW et al : Finite element analysis of fin−type implant fixtures. J Kor Oral Maxillofac Surg 29:14−25, 2003.

5. Siegele D, Soltesz U : Numerical investigations of the influence of implant shape on stress distribution in the jaw bone. Int J Oral Maxillofac Implants 4:333, 1989.

6. Ji S, Lee SH, Kim SG, Kim BO : Finite element analysis of the interface between the native bone and its surrounding regenerated bone. J Kor Asso Maxillofac Plast Reconstr Surg 26:24−33, 2004.

7. Kim BO, Hong KS, Kim SG : Finite element analysis of cylinder type implant placed into regenerated bone with type Ⅳ bone quality. J Kor Oral Maxillofac Surg 30:331−338, 2004.

8. Kim SH, Kim SG, Shim HS, Kim BO : Finite element analysis of the junction the regenerated bone adjacent to screw type implant and its surrounding native bone. J Kor Asso Maxillofac Plast Reconstr Surg 26:427−434, 2004.

9. Ciftci Y, Canay S : The effect of veneering materials on stress distribution in implant-supported fixed prosthetic restorations. Int J Oral Maxillofac Implants 15:571-582, 2000.

10. Kregzde M : A method of selecting the best implant prosthesis design option using three-dimensional finite element analysis. Int J Oral Maxillofac Implants 8:662-673, 1993.

11. 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 38:491-496, 1993.

12. 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 10:51-57, 1995.

13. Teixeira ER, Sato Y, Akagawa Y, Shindoi N. A comparative evaluation of mandibular finite element models with different lengths and elements for implant biomechanics. J Oral Rehabil 25:299-303, 1998.

14. 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 13:163-174, 1998.

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 14:384-391, 1999.

16. Quirynen M, Naert I, van Steenberghe D : Fixture design and overload influence marginal bone loss and fixture success in the Brånemark system. Clin Oral Implants Res 3:104-111, 1992.

17. 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 1:7-11, 1998.

18. 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 12:648-657, 2001.

Abstract

내부 임프란트/지대치 연결 임프란트의 이식 깊이에 따른 유한요소법적 응력 분석

조선대학교 치과대학 구강악안면외과학교실
조선대학교 치과대학 치주과학교실*
조선대학교 치과대학 보철학교실**
김수관, 최동국, 김병옥*, 오상호**

이 연구는 임프란트 식립시 골내 이식 깊이와 내부 임프란트/지대치 연결 임프란트 종류에 따른 3차원적 유한요소법을 통해 골, abutment screw, 임프란트에서 응력 분포를 비교 평가하여 임상적 유용성을 평가하는 데 그 목적이 있다.

모델링은 골 구조중 외부는 2mm의 두께를 갖는 치밀골로 처리하였고, 그 내부는 해면골로 처리하였다. 임프란트 고정체의 이식 깊이는 cover screw의 상단이 상단 피질골에 위치되도록 하였다. 하중조건 A는 치관 중심부에 200N의 수직 하중과 30도 경사 하중을, 하중조건 B는 협측 교두에 200N의 수직 하중과 30도 경사하중이 작용되도록 하여 stress를 분석하여 다음과 같은 결과들을 얻었다.

1. 골의 응력을 비교하면 하중점 A와 B점에 수직으로 작용하는 경우와 30도 경사 하중으로 작용하는 경우에 골에 가장 응력이 적게 받는 형태는 모델 5이었다.
2. crown인 경우 등가응력이 가장 적은 응력을 받는 implant/abutment connection은 하중점이 A일 때 하중이 90도로 수직으로 작용하는 경우와 30도로 경사 하중으로 작용하는 경우에 공통으로 모델 5이었다.
3. abutment screw의 경우는 등가응력을 비교하면 하중점 A와 B점 그리고 하중이 30도로 경사 하중으로 작용하는 경우에 가장 응력이 적게 받는 경우는 모델 5이고, 가장 응력이 많이 받는 경우는 모델 2이었다.
4. 임프란트의 경우는 하중점 A와 B점에서 하중이 수직으로 작용하는 경우와 30도로 경사 하중으로 작용하는 경우에 가장 응력이 적게 받는 경우는 모델 5이었다.
5. 대체적으로 전체적인 경향을 분석해 보면 응력이 적게 받는 모델의 형태는 1-->2-->3-->4-->5 형태로 모델 5로 갈수록 적은 응력을 보였다. 또한 같은 하중점에 하중이 수직으로 작용하는 경우보다 하중이 30도로 경사 하중으로 작용하는 경우에 더 많은 응력이 작용하고 있음을 알 수 있었다.

이러한 결과를 토대로 보면 임상적으로 치조제 상방으로 과도하게 2mm 상방으로 위치시키는 경우는 응력의 분산 관점에서 불리한 영향을 가지므로 적응증이 되는 특별한 경우를 제외하고는 그 사용을 제한하는 것이 필요하리라고 생각된다.