Three-dimensional FEM Analysis of Implant for Controlling Stress Concentration on Surrounding Bone

Jun Shimada, Kiyoshi Tajima and Hiroshi Kakigawa

Department of Materials Science (Director: Prof. Yoshio Kozono)

Kyushu Dental College, Kitakyushu, Japan

Accepted on December 25, 2000.

Abstract

In order to investigate the mechanical effects of the design of osseointegrated dental implant on the surrounding bone, the present study adopted more realistic approach by 3D FEM for stress analysis. An osseointegrated, screw type implant for molar region was selected, and the standard model and modified models in which the cusp inclination was increased to 40°, the abutment inclination was increased to 20° or the occlusal plane was buccolingually expanded were constructed. The results showed that the alteration of the configuration of superstructure, the occlusal contact positions or the occlusal relief variously changed the aspects of the stress distribution. However, any arrangement of these factors could not drastically relieve the stress concentration at the marginal crest of interfacial cortical bone so long as the occlusal contacts were limited to two points. An additional contact on the distal fossa to form three-point occlusal contacts could significantly reduce the maximum principal stresses and relieve the stress concentration.

It was reconfirmed through the 3D FEM analyses that the proper design of superstructure, occlusal contour and balanced disposition of occlusal contacts would minimize the stress concentration on surrounding bone in centric occlusion, lateral sliding or mastication of foods and contribute to the good performance of implant restorations.

Key words: 3D finite element method / Osseointegrated implant / Stress concentration / Occlusal contact / Superstructure

Introduction

One of the major problems in osseointegrated dental implant is a risk of mechanical damage to the surrounding bone. As the implant prosthesis is embedded in a jaw bone without any shock absorber corresponding to the periodontal ligament, the stress distribution on the surrounding bone has attracted a great deal of attentions. Many studies on this matter have been made by means of mechanical testing using load cell or strain gage with or without existence of foods¹⁻³⁾, geometric analysis of moment or vector⁴⁻⁸⁾, and computer simulation by FEM⁹⁻²¹⁾. The FEM has a useful merit that can make a model of the implant

and surrounding jaw bone with their components based on clinical shapes and material properties. Many FEM analyses were reported on the parameters of osseointegration such as size of implant and fixture, occlusal force direction⁹⁾, quality of bone¹⁰⁾. However, these analyses were made two-dimensionally on a given cross-sectional plane. The simulation model had to be simplified and the loading direction was selected for convenience as horizontal, vertical or oblique. Such simulations cannot entirely reproduce the mechanical behaviors of the implant.

The recent development of the computer-aided technology made it possible to reconstruct the three-dimensional (3D) image from successive two-dimensional (2D) images. Then some FEM stress analyses were made on the 3D simulation model focusing on the variables such as degree of osseointegration¹¹⁾ abutment angle¹²⁾, situation of bone^{13,14)} and connection of implant and tooth¹⁵⁾. Even in the 3D analyses, however, the loading and stress detection have so far been limited on a particular cross-sectional plane.

The present study adopted more realistic approach for 3D FEM stress analysis on an osseointegrated screw type implant. The authors previously developed the system to reconstruct the three-dimensional image of mandible from the two-dimensional CT image using personal computer²²⁾. Furthermore, a software is now available to treat simultaneously applied multiple forces in different directions. Utilizing these softwares, the stress distributions on surrounding bone of implant were evaluated by one to three simultaneous loading to minimize the stress concentration in association with the design of the configurations of abutment and superstructure.

Materials and Methods

The three-dimensional (3D) FEM analysis was carried out on the osseointegrated screw type molar implant and surrounding bone tissues in mandible using a personal computer (PC 9821 V13, NEC; 133 MHz, RAM64MB) for modeling and analyzing.

For modeling the mandibular bone, a digitized parasagittal image of human adult mandible was reconstructed from an image of computed tomography (CT) and the outlines of cortical and cancellous bone were traced on it. These outlines were converted into IGES file (Design LT 97, Ray, Co., Japan) and exported to FEM software (ANSYS 5.6, ANSYS, Inc., USA). From the transferred lines simply showing the cross-sectional outlines of the components of bone, two areas were created consisting of the outer cortical bone and the inner cancellous bone (Fig. 1). Then these areas were extruded by 22 mm and made into a 3D solid model as a piece of bone.

Implant fixture was designed to be a screw type, 13 mm in length and 4 mm in diameter. Abutment was 4 mm in height and 4.4 mm in diameter, of which head and lower pin were connected to crown and fixture, respectively. Both the fixture and abutment were assumed to be made of titanium.

It has been advocated that the implant crown in the posterior region should be sized to premolar to lighten the load imposed on its occlusal plane⁶⁾. According to this clinical

Table 1 Properties of materials used for model

Material	Young's modulus (MPa)	Poisson's ratio	Reference
Cortical bone	14,700	0.3	Moroi <i>et al.</i> ⁽¹⁷⁾
Cancellous bone	490	0.3	Moroi <i>et al.</i> ⁽¹⁷⁾
Titanium	110,000	0.35	Sertgoz ⁽¹⁸⁾
Porcelain	68,900	0.28	Sertgoz ⁽¹⁸⁾

requirement, the crown was made to simulate premolar by 7.6 mm in buccolingual width and 7.4 mm in mesiodistal width. The origin of the crown was drawn on the 3D graphic software (Shade, Expression Tools, Inc.) which could create cusps, ridges, fossae, and lateral contour similar to natural tooth crown. It was then exported to FEM software. After skinning, it was made into a 3D solid crown composed of a ceramometal structure with undulating occlusal plane involving two cusps and central groove.

These implant components were connected with each other, holding the interface in common, and embedded in the bone solid model. The interface between fixture and bone was constructed in the same way, assuming that the osseointegration was completely achieved (Fig. 1).

The individual properties of the materials such as Young's modulus and Poisson's ratio were cited from the published data^(17,18) as shown in Table 1.

After the solid modeling was completed, each part of the model was divided into a number of segments (elements). Adjacent elements were connected at specific points (nodes) on their common boundaries. Elastic strains at the nodes of any element were calculated from those of the adjacent elements using their moduli⁽⁹⁾. The element type used in this study was tetrahedral structural solid defined by ten nodes. More detailed meshing was set up in the region closer to the bone-implant interface to get precise data as shown in Fig. 2. The total number of the element was approximately 10,500 with 15,500 nodes.

The stress analyses were performed for the following simulated models. Figs. 3 to 6 show the examples of the stress distributions on the surrounding bone, in which the color going closer to red exhibits larger stress.

I. Simulation for one point loading

The FEM models were constructed for four variables, i.e. cuspal inclination, abutment inclination, vertical offset and buccal offset⁽⁷⁾ (Fig. 7). In each of them, one concentrated load of 100 N was applied perpendicularly to the inner incline of buccal cusp. The loading vector was fixed in one cross-sectional plane in every model so as to make it possible to compare with the geometric analyses⁽⁷⁾.

II. Simulation for two occlusal contacts

In order to examine the effects of the positioning of occlusal contacts on stress distribution, several combinations of two occlusal contact areas were simulated on the standard type

Fig. 1 Solid models of bone and implant components.

Fig. 2 3D finite element model.

Fig. 3 An example of stress distribution on surrounding bone.

Fig. 4 Magnified view of Fig. 3 around interfacial bone.

Fig. 5 Mesiodistally cross-sectioned view of Fig. 3.

Fig. 6 Emphasized expression of small stress in cancellous bone.

Fig. 7 Models for four variables.
 (a) Cuspal inclination
 (b) Abutment inclination
 (c) Vertical offset
 (d) Buccal offset

model of implant^{4-6, 23-25)}. The concerned regions of the occlusal plane were divided into small sections having same area (0.4×0.4 mm) as shown in Fig.8. One contact area was selected from the inner incline (a to e) or tip (t) of buccal cusp and the other the distal fossa (m to o) to form two contacts. The contacts were displaced mesiodistally by sifting one contact from a to e or t against the other given contact m, and buccolingually by shifting one from c to

Fig. 8 Loading areas for two occlusal contacts.

Fig. 9 Four models simulating clinical modifications.

- (a) Standard model
- (b) 40° cusp model: cuspal inclination was increased to 40°
- (c) 20° abutment model: abutment inclination was increased to 20°
- (d) Expanded occlusal surface model: occlusal surface was buccolingually expanded by 2 mm

h against m or from m to o against f. The applied load was the surface load on the selected section. Therefore, the normal direction or the direction of occlusal force was dependent of the occlusal plane contour.

The applied total load was defined as 100 N²⁶⁾ and it was divided into two surface loads of 312.5 MPa on each section. The stress on the surrounding bone was expressed by maximum principal compressive and tensile stresses.

The stress analyses were also carried out for three other modified models simulating frequently utilized structures in clinical practice involving those in which the cusp inclination was increased to 40 degrees, the abutment inclination was increased to 20 degrees and occlusal plane was buccolingually expanded by 2 mm (Fig.9). These were abbreviated as 40° cusp, 20° abutment and expanded occlusal plane model, respectively.

Fig.10 Path definition along marginal crest of interfacial bone.

Fig.11 Loading areas for three occlusal contacts.

For more detailed analysis of stress concentration, path analysis was employed, which pursued the stress along the marginal crest of interfacial bone shown in Fig.10.

III. Simulation for three occlusal contacts

An additional contact was given to the mesial fossa (p) for every model mentioned above as shown in Fig.11. The applied total load was equally 100 N consisting of 312.5 MPa to the contact in cusp area and 156.25 MPa to each of the two contacts in fossa. All the analyzing procedures followed those for the two occlusal contacts.

Results

The results of the 3D FEM analysis were displayed by maximum principal stress and von Mises stress distributions. As shown in the examples in Figs.3 to 5, similar patterns of stress distribution were revealed mainly in cortical bone layer around fixture and stress concentration at the buccal, marginal crest of interfacial bone for all the models. In cancellous bone, on the other hand, conspicuous stress distribution was found around implant apex although the stress level was extremely smaller than that of cortical layer (Fig.6).

I. One point loading

Figs.12 and 13 show the changes in maximum von Mises stress according to the variables when a concentrated load was applied to the inner incline of buccal cusp. The maximum stress increased as the cuspal inclination increased (Fig.12), the vertical offset of abutment was higher (Fig.13) and the buccal offset was shifted more buccally (Fig.13) while it decreased with increase in the abutment inclination (Fig.12). The cuspal inclination was the largest affective variables of all.

II. Loading by two occlusal contacts

Fig.14 shows the maximum principal stresses in compression and tension when the buccal contact in the two occlusal contacts was displaced mesiodistally. The compressive stress increased when the contacts shifted from the central position (cm) to the outer positions (bm

Fig. 12 Maximum von Mises stress in relation to cuspal inclination or abutment inclination.

Fig. 13 Maximum von Mises stress in relation to vertical offset or buccal offset.

Fig. 14 Maximum principal stresses in relation to mesiodistal displacement of occlusal contacts for standard model.

Fig. 15 Maximum principal stresses in relation to buccolingual displacement of occlusal contacts for standard model.

and dm) and decreased again to at the both ends (am and em). The cusp tip contact (tm) exhibited maximum stresses on the lingual interface with slightly smaller compressive stress and much larger tensile stress than the others. Fig. 15 also shows the changes in the maximum principal stresses for the buccolingual displacement of the occlusal contacts. The stresses increased as the occlusal contacts shifted toward the higher positions of the cuspal inclined plane (hm→cm) as well as the lingual positions of distal fossa (fm→fn and fo). The maximum compressive stress appeared at the buccal, marginal crest of interfacial bone and the maximum tensile stress at the lower boundary of the cortical bone layer on the same side or the marginal crest of interfacial bone on the opposite side (Fig. 16).

The maximum principal stresses for the modified models of 40° cusp, 20° abutment and expanded occlusal plane models were shown in Figs. 17 to 19, respectively. Their changing patterns in relation to the mesiodistal displacement of the occlusal contacts were similar to

Fig.16 Maximum compressive stress (MCS) and tensile stress (MTS).

Fig.17 Maximum principal stresses for 40° cusp model.

Fig.18 Maximum principal stresses for 20° abutment model.

Fig.19 Maximum principal stresses for expanded occlusal plane model.

that of the standard model. However, the stress levels were different from each other. The 40° cusp model showed the largest maximum principal stresses in average and the expanded occlusal plane model the next, and the standard model, the 20° abutment model, in that order.

Figs.20 to 23 show the von Mises stresses representing stress concentrations as the results of path analyses along the marginal crest of interfacial bone for the standard and modified models. Each of the models exhibited individually characteristic aspect of the stress concentration. In the standard model, predominant stress concentration was observed in narrow buccal area being less affected by the displacement of the occlusal contacts except for the tm contact showing quite opposite stress distribution. The 40° cusp model showed wide deviations of the stress distribution along the marginal crest as well as by the displacement of the occlusal contacts and sharp stress concentrations in dm and em contacts. In the 20° abutment model, the stress distribution along the marginal crest was relatively flat at lower

Fig. 20 Path analysis along marginal crest of interfacial bone for standard model.

Fig. 21 Path analysis along marginal crest of interfacial bone for 40° cusp model.

Fig. 22 Path analysis along marginal crest of interfacial bone for 20° abutment model.

Fig. 23 Path analysis along marginal crest of interfacial bone for expanded occlusal plane model.

level and the stress concentration was the smallest. The expanded occlusal plane model showed quite similar aspects of stress concentration to the standard.

III. Loading by three occlusal contacts

When the system of three occlusal contacts having additional contact on the mesial fossa was adopted for the standard and modified models, the maximum principal stresses and Von Mises stresses were detected as shown in Figs. 24 to 31. In general, the maximum principal stresses were lowered in any model. Especially the maximum compressive stresses significantly decreased and the difference between the compressive and tensile stresses was reduced. The stress concentrations were also relieved. In the standard and the expanded occlusal plane models, the stress levels along the marginal crest of interfacial bone were fairly evened and the predominant stress concentrations almost disappeared except for the contact position of tm. The stresses in the 40° cusp model were also lowered although the appearance of the stress concentration was still kept unchanged. In the 20° abutment model, the stresses were completely flattened.

Fig.24 Maximum principal stresses for standard model with three occlusal contacts.

Fig.25 Maximum principal stresses for 40° cusp model with three occlusal contacts.

Fig.26 Maximum principal stresses for 20° abutment model with three occlusal contacts.

Fig.27 Maximum principal stresses for expanded occlusal plane model with three occlusal contacts.

Discussion

In the stress analyses of bone, it has been reported that the bone conditions such as thickness, density and internal structure of the cortical bone might be the significant factors affecting the detected stress distribution^{10,13,14)}. Since these variables are all different among individuals or sections of a bone, it is difficult to universally evaluate the effects of implant design on the interfacial tissue through stress analyses on a seriously reconstructed bone model. The present 3D FEM analysis, therefore, tried to standardize the solid model, in which a 2D graphical area derived from one cross-sectional CT image of an adult mandible was extruded and converted into a 3D bone piece. This approach could make a model with entirely homogeneous bone structure and eliminate unexpected deviations of data in association with such irregularities mentioned above.

As for the implant components involving fixture, abutment and superstructure, the

Fig. 28 Path analysis for standard model with three occlusal contacts.

Fig. 29 Path analysis for 40° cusp model with three occlusal contacts.

Fig. 30 Path analysis for 20° abutment model with three occlusal contacts.

Fig. 31 Path analysis for expanded occlusal plane model with three occlusal contacts.

conventional stress analyses have commonly dealt with one body unifying all together. In this study, they were individually modeled as the separate components to simulate more naturally the actual situation. The comparative examination for both the simulations in the preliminary experiment showed that the unified implant model produced significantly larger maximum stress on the interfacial tissue than the separated model. It was probably because in the unified model all the implant components behaved as a bulk although their responses to the applied force were inherently different.

In dental implant, the phenomenon of stress concentration against particular surrounding tissue is structure sensitive and it is important to take it into consideration in designing. When a load was applied to the cuspal inclined plane of the superstructure, the stress detected on the surrounding tissue was most widely changed by the change in the degree of cuspal inclination and least by that of abutment inclination. The results agreed with the two-dimensional geometric calculations by Weinberg (1995)⁷⁾, suggesting that the

configuration of the superstructure would be the prime factor affecting the distribution and magnitude of the stresses.

In order to diminish lateral forces which are generally considered harmful to implant and/or interfacial tissue, two-dimensional vector analyses have so far been designed for equilibrium of occlusal force components.⁴⁻⁶⁾ The design of the occlusal contacts may be regarded as an important factor that determines the vector of occlusal forces for geometric solution. Pokorny *et al.* (1980)²⁷⁾ classified the standard occlusal contacts into A-, B- and C-types (Fig. 32). Mcdevitt *et al.*

Fig. 32 A-, B- and C-contact position, classified by Pokorny *et al.*²⁷⁾

(1997)²⁵⁾ described that two inclined plane contacts on supporting cusp inclines which converge occlusally, or on the side walls of a fossa which converge to its base, had the effect of canceling each other out. For this reason they demonstrated that such contacts as A & B; B & C; or A, B & C contributed to occlusal stability. Weinberg (1998)⁸⁾ hypothesized that buccal and lingual component lines of force (generated by the cusp incline surfaces) produce a vertical resultant line of force that is biomechanically favorable. However, there were some opposite opinions that this is only correct theoretically because such repeatably precise fit is unattainable clinically²⁸⁻³¹⁾. If the occlusal contacts slightly change in position or size, the occlusal force will affect not only the buccolingual stress components within a fixed cross-sectional plane but also the three-dimensional, multiple components because the occlusal plane involves complex undulation in every direction. Such a situation may be easily produced in the practical mastication, in which intensive clenching or mastication will lead to displacement or enlargement of the occlusal contacts³²⁻³⁴⁾. Thus the conventional 2D analysis cannot sufficiently pursue the stress transmission and it is necessary to adopt the 3D analysis for more practical improvement in the stress distribution in association with the implant.

When two loads were simultaneously applied to the superstructure corresponding to B- and C-contacts on the basis of these concepts in 3D FEM analysis, only a slight displacement of the contact on a given cusp caused measurable change in the stress generation. The mesiodistal displacement of the B-contact (am→bm→cm→dm→em in Fig. 8) showed no definite tendency of the change in stress values (Fig. 14). The lingual displacement of the C-contact (fm→fn→fo in Fig. 8), on the other hand, accelerated the buccal stress (Fig. 15) against the expectation that it would have to theoretically cancel the buccal stress of B-contact. These results indicate that the disposition of the occlusal contacts may lose their balance and make unexpected force and twisting torque around the long axis of implant. It seems to be a significant cause of the stress concentration on the surrounding bone. In general, the failure in equilibrium of occlusion induced stress concentration. Furthermore, it

was found that even the cusp-fossa contact (t and m in Fig.8) in the standard pattern which was considered biomechanically favorable²⁵⁾ yielded lingual stress almost equal in magnitude to those of other contacts. Morikawa (1994)³⁵⁾ reported from the occlusal force measurements that the occlusal force on the lower first molar was detected toward lingual direction in some cases and buccal in other cases. He also demonstrated that the direction of occlusal force might be determined by the cuspal inclination and the position of occlusal contacts in the centric occlusion regardless of the existence of foods. These findings together with the results in this study suggest that the direction of stress through implant can be controlled by the occlusal contour and the position of occlusal contacts.

Comparison of the implant models employed involving standard, 40° cusp, 20° abutment and expanded occlusal plane models, revealed that the maximum stress levels detected on the surrounding bone were different among the models (Figs.14 and 17 to 19). It seemed to reflect the variation in the moment of force caused by the difference in configuration of the superstructure. On the other hand, the changing pattern of the maximum stress in relation to the displacement of the occlusal contacts from am to em was generally similar for all the models. It is difficult to explain the clear aspects of the stress concentration from these results. The path analysis of the stress along the marginal crest of interfacial bone could demonstrate a characteristic relationship between the maximum stress and the stress concentration. As seen in Figs.20 to 23, distinct stress concentration appeared in the model which showed larger maximum stress. The expanded occlusal plane model exhibited large stress concentration similarly to the standard model and it was less affected by the displacement of the occlusal contacts. The 40° cusp model with a predominant occlusal relief also showed larger stress concentration than the other models. In addition the aspect of the stress concentration varied with displacement of the occlusal contacts. It suggests that the lateral slide of the contact points in this type of occlusal design may induce fluctuation of the stress concentration. It was also confirmed that the whole stress level might depend on the configuration of the superstructure and the difference between the top and bottom in the stress distribution might be significantly changed by the contact point pattern as well as the occlusal relief. This is partly supported by an in vivo report on horizontal bending moment on implant induced by lateral force²⁾.

Since occlusal force varies in a wide range of magnitude by clenching or mastication^{26, 36-38)}, it is important for investigating the damage of the surrounding bone through implant to compare the condition of stress concentration rather than to observe each stress value. Thus the present study performed the path analysis along the marginal crest of interfacial bone, in some point of which an extreme stress concentration commonly appeared in all the cases. The results showed that the alteration of the configuration of the superstructure, the occlusal contact points or the occlusal relief produced various aspects of stress concentration at the crest. Any arrangement of these factors could not drastically relieve the stress concentration so long as the occlusal contacts were limited to two points. The stress concentration may potentially cause a risk of the crestal bone loss. As an attempt for improving such situations,

therefore, an additional contact point was given to the mesial fossa within C-contact. This three-point occlusal contact system could successfully reduce the maximum stress. The pass analysis showed fairly even stress distribution along the marginal crest of interfacial bone, and the significant stress distribution disappeared in all the models. In the 2D stress analysis on a given plane, it may not always be impossible to attain an equilibrated occlusion with implant by arranging the B- and C-contacts. In the practical situation, however, any cusp has three-dimensional complex inclined plane and it is difficult to completely cancel the B-contact by a single C-contact. The third contact on the cusp fossa seemed to establish a condition of balanced occlusal contacts to restraint lateral force or twisting torque of B-contact.

It was reconfirmed through the 3D FEM analyses that the proper design of superstructure, occlusal contour and balanced disposition of occlusal contacts would minimize the stress concentration on surrounding bone in centric occlusion, lateral sliding or mastication of foods and contribute to the good performance of implant restorations.

Conclusion

Three-dimensional FEM analysis of dental implant was performed to examine the mechanical effects of the configuration of superstructure and occlusal contacts on the surrounding bone. An osseointegrated, screw type implant for molar region was selected, and the standard model and modified models in which the cusp inclination or the abutment design was changed, or the occlusal plane was buccolingually expanded were constructed. The results obtained were as follows.

(1) When one concentrated load was applied to the inner incline of buccal cusp, the stress on the surrounding bone was most affected by the change in the cuspal inclination and least by the abutment inclination.

(2) In the case of the two occlusal contacts at the buccal cusp and the distal fossa, the maximum principal stresses were fluctuated by a slight displacement of the occlusal contacts.

(3) When the two occlusal contacts were displaced in the modified models, similar changing pattern to the standard was observed in relation to the positions of the contacts. However, the stress levels were different among the models; the 40° cusp model showed the largest level and the next the expanded occlusal plane model, the standard and the 20° abutment model, in that order.

(4) The stress concentration was primarily detected on the buccal, marginal crest of interfacial cortical bone. The aspects of the stress concentration was significantly changed by the configuration of superstructure and the positions of occlusal contacts

(5) An additional contact on the distal fossa to form three occlusal contacts could drastically decrease the maximum principal stresses and relieve the stress concentration.

Acknowledgement

The authors are very deeply indebted to Prof. Yoshio Kozono for his invaluable advice and

discussions throughout this investigation, and also wishes to express great gratitude to Prof. Yasunari Uchida and Prof. Masamichi Terashita for their helpful advices. The authors express deep thanks to Dr. Yuki Nagamatsu, Ms. Katsuyo Nakamura, Dr. Maho Fukuizumi and Ms. Satoko Matsunobu for their continued encouragement.

This study was partly supported by the grant-in-aid of the Association of Supporters for Kyushu Dental College.

References

- 1) Kaukinen, J. A., Edge, M. J. and Lang, B. R.: The influence of occlusal design on simulated masticatory forces transferred to implant-retained prostheses and supporting bone. *J. Prosthet. Dent.* 76: 50-55, 1996.
- 2) Richter, E. J.: *In vivo* horizontal bending moments on implants. *Int. J. Oral. Maxillofac. Implants.* 13: 232-244, 1998.
- 3) Wang, L., Sadler, J. P., Breeding, L. C. and Dixon, D. L.: An *in vitro* study of implant-tooth-supported connections using a robot test system. *J. Biomech. Engineering.* 121: 290-297, 1999.
- 4) Misch, C. E. and Bidez, M. W.: Implant-protected occlusion. *Pract. Periodontics Aesthet. Dent.* 7: 25-29, 1995.
- 5) Takayama, T.: Biomechanical guidelines for upperstructure. In: *Osseointegration implant superstructure actual* (ed. by Komiyama, Y., Soejima, Y., Tanoguchi, K., Tamura, K., Nishikido, T. and Watanabe, F.). QDT Extra Issue. Quintessence, Tokyo, 1992, 26-38. (in Japanese)
- 6) Nishikido, T.: Superstructure and occlusion. In: *Osseointegration implant superstructure actual* (ed. by Komiyama, Y., Soejima, Y., Tanoguchi, K., Tamura, K., Nishikido, T. and Watanabe, F.). QDT Extra Issue. Quintessence, Tokyo, 1992, 68-72. (in Japanese)
- 7) Weinberg, L. A. and Kruger, B.: A comparison of implant/prosthesis loading with four clinical variables. *Int. J. Prosthodont.* 8: 421-433, 1995.
- 8) Weinberg, L.: Reduction of implant loading using a modified centric occlusal anatomy. *Int. J. Prosthodont.* 11: 55-69, 1998.
- 9) Holmgren, E. P., Seckinger, R. J., Kilgren, L. M. and 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 Implantology* 24: 80-88, 1998.
- 10) Clelland, N. L., Lee, J. K., Bimbenet, O. C. and Gilat, A.: Use of an axisymmetric finite element methods to compare maxillary bone variables for a loaded implant. *J. Prosthodont.* 2: 183-189, 1993.
- 11) Papavasiliou, G., Kamposiora, P., Bayne, S. C. and Felton, D. A.: 3D-FEA of osseointegration percentages and patterns on implant-bone interfacial stresses. *J. Dent.* 25: 485-491, 1997.
- 12) Clelland, N. C., Lee, J. K., Bimbenet, O. C. and Brantley, W. A.: A three-dimensional finite element stress analysis of angled abutments for an implant placed in the anterior maxilla. *J. Prosthodont.* 4: 95-100, 1995.
- 13) Papavasiliou, G., Kamposiora, P., Bayne, S. C. and Felton, D. A.: Three-dimensional finite element analysis of stress-distribution around single tooth implants as a function of bony support, prosthesis type, and loading during function. *J. Prosthet. Dent.* 76: 633-640, 1996.
- 14) Holmes, D. C. and Loftus, J. T.: Influence of bone quality on stress distribution for endosseous implants. *J. Oral Implantology* 23: 104-111, 1997.
- 15) Sato, K., Kusakari, H. and Miyakawa, O.: Three dimensional finite element analysis of bone around dental implants in posterior mandibular region. -Biomechanics of implant connections-. *J. Jpn. Prosthodont. Soc.* 40: 682-694, 1996. (in Japanese)

- 16) Clelland, N. L., Ismail, Y. H., Zaki, H. S. and Pipko, D.: Three-dimensional finite element stress analysis in and around the screw-vent implant. *Int. J. Oral Maxillofac. Implants* 6: 391-398, 1991.
- 17) Moroi, H., Okimoto, K., Moroi, R. and Terada, Y.: Numeric approach to the biomechanical analysis of thermal effects in coated implants. *Int. J. Prosthodont.* 6: 564-572, 1993.
- 18) Sertgoz, A.: Finite element analysis study of the effect of superstructure material on stress distribution in an implant-supported fixed prosthesis. *Int. J. Prosthodont.* 10: 19-27, 1997.
- 19) Chen, J., Lu, X., Paydar, N., Akay H. U. and Roberts, W. E.: Mechanical simulation of the human mandible with and without an endosseous implant. *Med. Eng. Phys.* 16: 53-61, 1994.
- 20) Shindoi, N.: Development of a three-dimensional finite element model with *in vivo* implant-bone interface structure. *J. Hiroshima Univ. Sch. Dent.* 31: 55-71, 1999. (in Japanese)
- 21) Akpinar, I., Anil, N. and Parnas, L.: A natural tooth's stress distribution in occlusion with a dental implant. *J. Oral Rehab.* 27: 538-545, 2000.
- 22) Shimada, J., Tajima, K., Kakigawa, H., Nagamatsu, Y., Naito, T., Yokota, M. and Kozono, Y.: Three-dimensional image reconstruction of jaw bone with a personal computer. *J. Kyushu Dent. Soc.* 53: 325-331, 1999. (in Japanese)
- 23) Horiguchi, Y.: Clinical anatomy of occlusion. Ishiyaku Publishers Inc., Tokyo, 1991. (in Japanese)
- 24) Fukushima, S., Furuya, R. and Hirai, T.: Clinical occlusion. Ishiyaku Publishers Inc., Tokyo, 1992. (in Japanese)
- 25) Mcdevitt, W. E. and Warreth, A. A.: Occlusal contacts in maximum intercuspation in normal dentitions. *J. Oral Rehabil.* 24: 725-734, 1997.
- 26) Shiota, M. and Ohno, S.: Clinical study of occlusal forces on dental implants. *J. J. Oral Bio.* 64: 405-412, 1997.
- 27) Pokorny, D. K. and Blake, F. P.: Principles of occlusion. 1st ed. Denar Corp., Anaheim, 1980, 39.
- 28) Grasso, J. and Sharry, J.: The duplicability of arrow-point tracing in dentulous subjects. *J. Prosthet. Dent.* 20: 106-115, 1968.
- 29) Kantor, M., Silverman, S. and Garfinkel, L.: Centric relation recording techniques: A comparative investigation. *J. Prosthet. Dent.* 28: 593-600, 1972.
- 30) Calagna, L., Silverman, S. and Garfinkel, L.: Influence of neuromuscular conditioning on centric relation registrations. *J. Prosthet. Dent.* 30: 598-604, 1973.
- 31) Celenza, F. V.: The centric position: Replacement and character. *J. Prosthet. Dent.* 30: 591-598, 1973.
- 32) Kumagai, H., Suzuki, T., Hamada, T., Sondang, P., Fujitani, M. and Nikawa, H.: Occlusal force distribution on the dental arch during various levels of clenching. *J. Oral Rehabil.* 26: 932-935, 1999.
- 33) Sakaguchi, R. L., Anderson, G. C. and DeLong, R.: Digital imaging of occlusal contacts in the intercuspation position. *J. Prosthodont.* 3: 193-197, 1994.
- 34) Murata, Y.: The influence of occlusal height of a cast crown on occlusal contact area. 1. Bite force in the silicone-black method. *J. Jpn. Prosthodont. Soc.* 32: 601-609, 1988. (in Japanese)
- 35) Morikawa, A.: Investigation of occlusal force on lower first molar in function. *J. J. Oral Bio.* 61: 250-274, 1994. (in Japanese)
- 36) Kikuchi, M., Korioto, T. W. and Hannam, A. G.: The association among occlusal contacts, clenching effort, and bite force distribution in man. *J. Dent. Res.* 76: 1316-1325, 1997.
- 37) Suzuki, T., Kumagai, H., Yoshitomi, N., Minakuchi, S., Watanabe, T., Uchida, T., Ishinabe, S., Sekita, T., Kobayashi, K., Kobayashi, K. and Nagao, M.: Clinical evaluation of measuring system of occlusal force. *J. J. Oral Bio.* 61: 437-445, 1994. (in Japanese)
- 38) Ida, H.: Evaluation of the degree of restoration of function with fixed molar bridges. *J. Jpn. Prosthodont. Soc.* 33: 682-695, 1989. (in Japanese)

インプラント周囲骨の応力集中に関する 三次元有限要素解析

嶋 田 純・田 島 清 司・柿 川 宏

九州歯科大学歯科理工学講座（指導：小園凱夫教授）

歯科用インプラントの上部構造の形態および咬合様式がインプラント周囲骨におよぼす影響を調査するため、3次元有限要素法を用いて、インプラント植立モデルに対する応力解析を行った。対象は下顎臼歯部の単独歯型インプラントとし、上部構造のバリエーション（アバットメント角度、咬頭傾斜角度、咬合面サイズ、水平・垂直オフセット）、また咬合接触のパターンをシミュレートし、周囲骨に加わる応力の大きさ、および応力集中の動態について比較検討したところ、以下の結論が得られた。

1) 集中荷重をかけた上部構造の各形態のうち、周囲骨の応力に最も影響を与えたのは、咬頭傾斜角度であり、最も影響が少なかったのはアバットメント角度であった。

2) 標準型モデルの咬合面上において、支持咬頭内斜面、遠心小窩の2カ所に面荷重の咬合接触を与えた場合、わずかな接触部位の移動によっても周囲骨の応力に変化がみられた。

3) 上部構造の各形態に2カ所の咬合接触を与え荷重を加えると、形態に応じて周囲骨に異なる値の応力が発生した。また、支持咬頭内斜面上で咬合接触部位を移動させると、いずれの形態においても応力の変化が起こった。

4) 応力集中は頰側の骨頂部辺縁皮質骨に発生し、上部構造の形態や咬合面の起伏、咬合接触の位置によって大きく影響を受けた。

5) 周囲骨への応力集中を抑えるために、小窩での咬合を増やし、3カ所の咬合接触としたところ、荷重量が同じであるにもかかわらず、いずれの形態においても、周囲骨における最大応力は減少し、応力集中は緩和された。

以上の結果から、適切な上部構造の設計、咬合面形態、バランスのとれた咬合接触の配置によって、周囲骨に加わる応力集中は可及的に抑えることができ、これらの操作がインプラント修復の成功に寄与するものであることが示唆された。