Influence of Implant Abutment Type on Stress Distribution in Bone Under Various Loading Conditions Using Finite Element Analysis

Heoung-Jae Chun, PhD[1]/Ha-Shik Shin, MS[2]/Chong-Hyun Han, DDS[3]/Soo-Hong Lee, PhD[4]

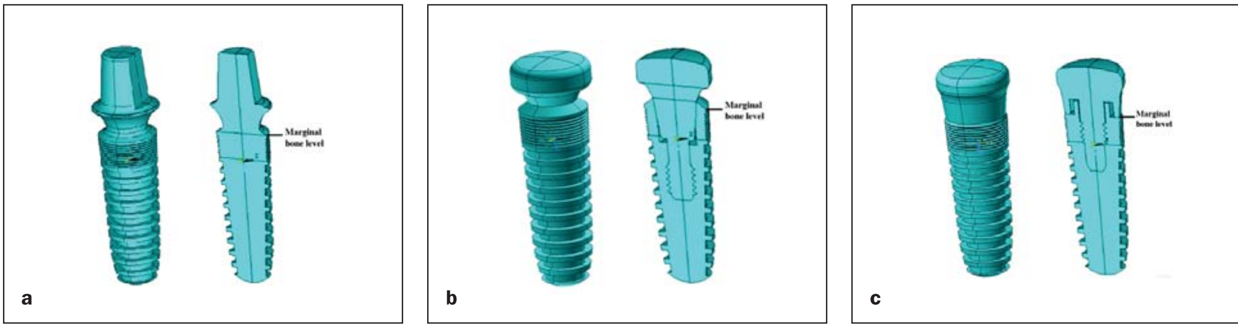
Purpose: *The purpose of this study was to investigate the effect of 3 different abutment types on the stress distribution in bone with inclined loads using finite element analysis.* ***Materials and Methods:*** *The 1-body, internal-hex, and external-hex implant systems were modeled to study the effect of abutment type on stress distribution in bone. The bone model used in this study comprised compact and spongious bone assumed to be homogeneous, isotropic, and linearly elastic.* ***Results:*** *In the case of the 1-piece implant, the load was transferred evenly not only in the implant system but also in bone. However, the maximum Von Mises stress generated in bone with the 1-piece implant was always higher than that generated with the internal-hex implant, regardless of load angle inclination. In the case of the internal-hex implant, the contact condition with friction between abutment and implant in the tapered joints and at abutment neck reduced the effect of bending caused by horizontal component of inclined load. The maximum Von Mises stress in bone was the highest for the external-hex implant.* ***Discussion:*** *It was found that the internal-hex implant system generated the lowest maximum Von Mises stresses for all loading conditions because of reduction of the bending effect by sliding in the tapered joints between the implant and abutment.* ***Conclusions:*** *It was concluded that abutment type has significant influence on the stress distribution in bone because of different load transfer mechanisms and the differences in size of the contact area between the abutment and implant.* (Basic Science) INT J ORAL MAXILLOFAC IMPLANTS 2006;21:195–202

Key words: *1-piece implants, external-hex abutment connection, finite element analysis, internal-hex abutment connection, stress distribution*

Poorly designed implants can create regions of increased stress in peri-implant bone and induce severe resorption, leading to gradual loosening and finally complete loss of the implant. Therefore, an analysis of stress distribution on bone with respect to different implant systems could contribute to the improvement of implant design. A number of studies have been conducted on stress in bone with different implant systems using the finite element method. Rieger and associates[1,2] reported that it is essential for the success of an osseointegrated implant that it be designed to distribute fracture stress on bone. Clelland and associates[3] performed 3-dimensional finite element analysis on the Screw-Vent implant (Zimmer Dental, Carlsbad, CA) and the bone surrounding the implant. Siegele and colleagues[4] studied stress distribution in peri-implant bone for different types of dental implants. The results demonstrated that different implant shapes lead to significant variations in stress distribution in the jawbone. Holmgren and coworkers[5] analyzed stresses in the jawbone for two particular implant shapes, the stepped cylindric and the straight implant shapes. From the results of their study, it was found that stress was more evenly dissipated in the jawbone with the stepped cylindric shape implant than with the straight shape.

[1]Associate Professor, School of Mechanical Engineering, Yonsei University, Seoul, Korea.
[2]Graduate Student, School of Mechanical Engineering, Yonsei University, Seoul, Korea.
[3]Professor, Department of Prosthodontics, Yong-dong Severance Hospital, Yonsei University, Seoul, Korea.
[4]Professor, School of Mechanical Engineering, Yonsei University, Seoul, Korea.

Correspondence to: Dr Heoung-Jae Chun, 134, Shinchon-Dong, Seodaemun-Ku, Seoul 120-749, Korea. Fax: +82 2 362 2736. E-mail: hjchun@yonsei.ac.kr



Figs 1a to 1c Schematic drawing showing different implant systems used in the analysis: *(a)* the 1-piece implant, *(b)* the internal-hex implant, and *(c)* the external-hex implant.

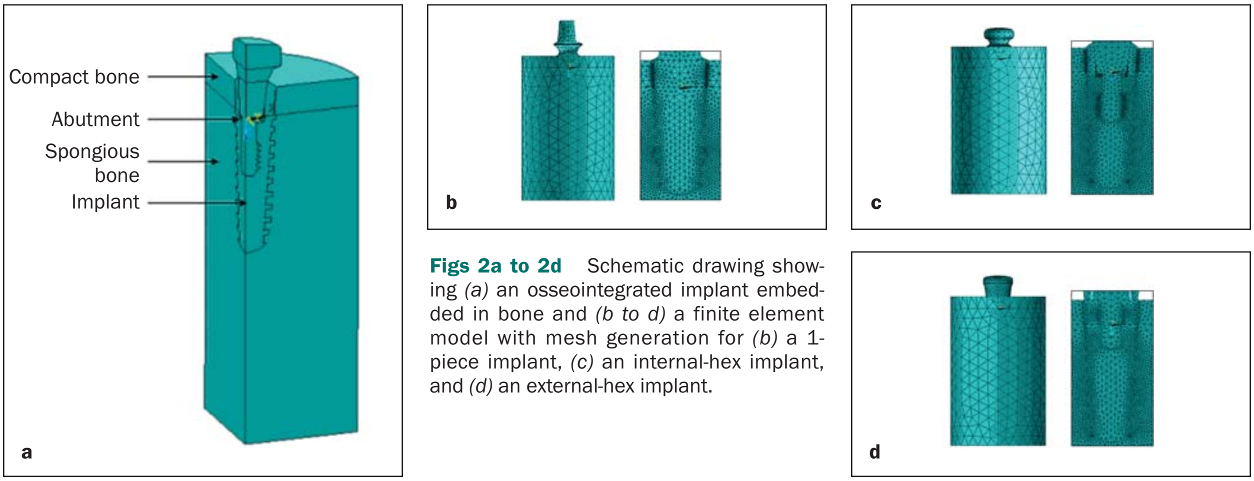
Chun and colleagues[6] performed finite element analysis on dental implants of various shapes to find an optimum thread shape producing more even stress distribution in jawbone. They concluded that the stress in bone was distributed more effectively as the length and diameter of the implant increased and as the pitch decreased. Pierrisnard and associates[7] compared locking-pin implants (ie, an implant design reinforced by 2 bicortical locking pins) to cylindric implants. They reported that the locking-pin implant had better early stability but that the cylindric implant was superior in stress distribution. Bozkaya and Muftu[8] analyzed the mechanics of tapered interference fits using a closed-form formula and finite element method to predict the contact pressure. They suggested that plastic deformation in the implant limited the increase in the pullout force that would have been otherwise predicted by higher interference values. The tapered fit provided a reliable connection method between the abutment and the implant.

Hansson[9] found that the peak bone-implant interfacial shear stresses generated by the conical implant-abutment interface were less than those produced by the flat-top interface. The implant with the conical interface can resist a larger axial load than the implant with the flat-top interface, and the peak interfacial shear stress level was affected by the way in which the axial load was distributed on the flat top and on the inner conus. Merz and Hunenbart[10] conducted 3-dimensional, nonlinear finite element analysis and compared the 8-degree Morse taper and the butt joint as connections between an implant and an abutment, with the loading condition modeled according to an actual test setup used for the dynamic long-term testing of dental implants. The results gave insight into the mechanics involved in each type of connection and were also compared to the experimental findings. The comparative results showed that conical abutment connections were superior mechanically and helped to explain their significantly better long-term stability in clinical applications.

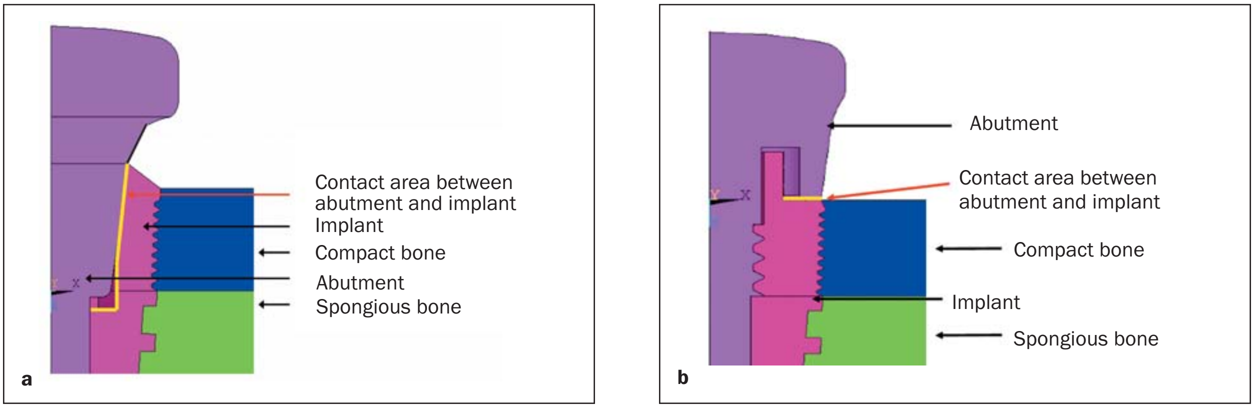
Previous studies have mainly focused on the effect of implant shape on stress distribution in bone and on the mechanics of implant and abutment connections for abutment loosening. However, the authors know of few studies focused on the relationship between stress distribution in bone and abutment type, especially among 1-piece, internal-hex, and external-hex implant systems. Since the load transfer mechanism of an implant can be altered significantly by the shape of the abutment, the objective of this study was to investigate the effect of 3 different abutment types on stress distribution in bone under vertical and inclined loads by performing finite element analysis.

MATERIALS AND METHODS

Three different implant systems produced by Warantec (Seoul, Korea) were modeled in this investigation to study the effect of abutment type on stress distribution in bone. In the analysis, the 3 implant systems are referred to as the 1-piece (Oneplant, OP-TH-S11.5), internal-hex (Inplant, IO-S11.5), and external-hex (Hexplant, EH-S11.5) implant systems. Oneplant is a 1-component implant system in which the implant and abutment consist of a single piece. Inplant is a 2-component implant system with a taper connection between the implant and abut-



Figs 2a to 2d Schematic drawing showing *(a)* an osseointegrated implant embedded in bone and *(b to d)* a finite element model with mesh generation for *(b)* a 1-piece implant, *(c)* an internal-hex implant, and *(d)* an external-hex implant.



Figs 3a and 3b Schematic drawing showing contact regions for *(a)* an internal-hex implant and *(b)* an external-hex implant.

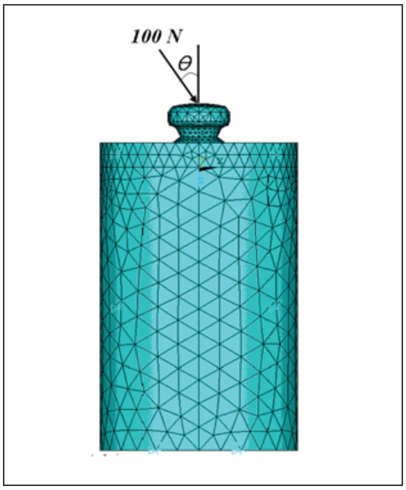
Table 1 Young's Modulus and Poisson's Ratio for Materials Used

Materials	Young's modulus (GPa)	Poisson's ratio
Titanium grade ELI (abutment)	113.8	0.34
Titanium grade IV (implant)	114.0	0.37
Compact bone	14.0	0.30
Spongeous bone	1.5	0.30

ment. Hexplant has a short external hex at the prospective connection with abutment. The Inplant and Hexplant are axi-symmetric, but the Oneplant is symmetric to a plane. The 3 implant systems with cross sections along the symmetric plane are shown schematically in Fig 1. Although they had different abutment types, all of them had the same standard implant shape, with a length of 11.5 mm and diameter of 4.3 mm. To simplify the analysis, the threads in the implant were modeled as circular rings instead of having a spiral configuration. The influence of the preload caused by tightening the abutment screw was neglected in this study because it may have small influence on the stress distributions in bone caused by different abutment connections.

Compact bone and spongious bone were assumed to be homogeneous, isotropic, and linearly elastic, and the material properties used in the 3-dimensional finite element analysis are shown in Table 1.[10] The thickness of the compact bone layer in the model was assumed to be 2 mm.

Figure 2a shows schematic drawings of the 1-piece, internal-hex, and external-hex implants embedded in bone. Figure 2b shows a schematic drawing of a finite element model with mesh generations. Eight-node iso-parametric elements were applied in the meshing procedure and analysis. Similar mesh densities were used for the bone for all 3 implant types in order to eliminate the effect of mesh density on the stress distribution in the bone.



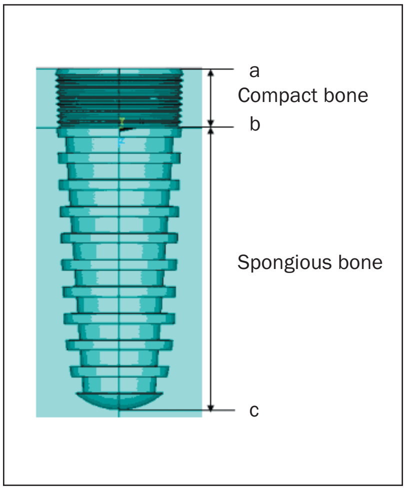


Fig 4 Schematic drawing showing applied loading directions and boundary conditions.

Fig 5 Schematic drawing of an implant embedded in both compact bone *(a to b)* and spongious bone *(b to c)*.

Figure 3 shows schematic drawings of the contact regions for the internal-hex and external-hex implants. Nonlinear contact with friction was assumed between abutments and implants in the tapered joints and at the abutment neck for the internal-hex implant and in the base of the hex and at the lower end of the abutment cap for the external-hex implant. In the analysis, the contact area was assumed to transfer only pressure and tangential frictional forces. These conditions were modeled by means of contact elements. Because of the different tapered angles of the abutment and implant for the internal-hex implant, initial contact was made only on the boundary where the abutment meets the implant in the tapered joints. Black lines in Fig 3 indicate the contact area between the abutment and implant. The friction coefficient[9] was assumed to be 0.5.

Figure 4 shows the boundary and loading conditions of the finite element analysis model used in this study. It was assumed that the interface between implant and bone was perfectly bonded and that displacements of the outer surface of bone in the model were fixed for all axes. Loading conditions considered in the analysis were a vertical load of 100 N and inclined loads of the same magnitude at 15, 30, and 60 degrees to the vertical axis of the abutment so as to compare the effect of moment caused by the horizontal component in the inclined force to the effect of vertical force.

Figure 5 is a schematic drawing of the implant in bone. To observe the variation of stress along the interface between implant and bone for both vertical and inclined loads, the most coronal section of the implant (2 mm) was assumed to be surrounded by compact bone, and remainder of the implant (9.5 mm) was assumed to be surrounded by spongious bone.

RESULTS

All results of numerical analysis are shown for Von Mises stresses. Eight different colors were selected for indication of specific values of Von Mises stress.

Figures 6, 7, and 8 show the stress distributions in bone with different abutment types and loading conditions. For the vertical loading condition, rotationally symmetric stress distributions were observed for the internal-hex and external-hex implants, since they were axi-symmetric. However, the interfacial stress distribution for the 1-piece implant was not rotationally symmetric, even when the vertical load was applied along the symmetric axis of the lower implant part, since the abutment is not symmetric. All 3 types of implant systems showed similar stress distribution in bone. However, as the angle of inclination of the load increased, the horizontal component of the inclined load also increased, which generated increase of moment and eventually increase of compressive load to a level higher than the compressive load generated by only the vertical component of the force at compact bone adjacent to the first microthread of the implant.

From the results, it was concluded that the magnitude of maximum Von Mises stresses increased as the inclination angle of applied load increased. Stresses were generated at the area of compact bone adjacent to the first implant microthread for all implant systems, and the difference in the level of maximum Von Mises stress in peri-implant bone increased noticeably. The external-hex implant generated the largest maximum Von Mises stress, whereas the internal-hex implant generated the smallest maximum Von Mises stress under all loading conditions, as shown in Fig 9. In the case of spongious bone, the maximum Von Mises stress was generated at the interface between the bottom of the

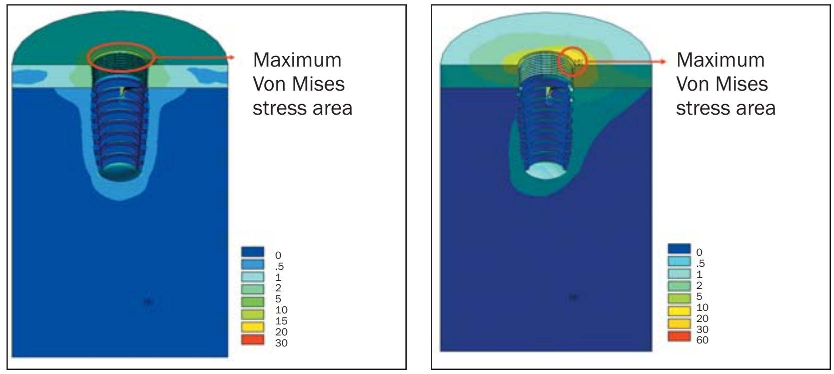
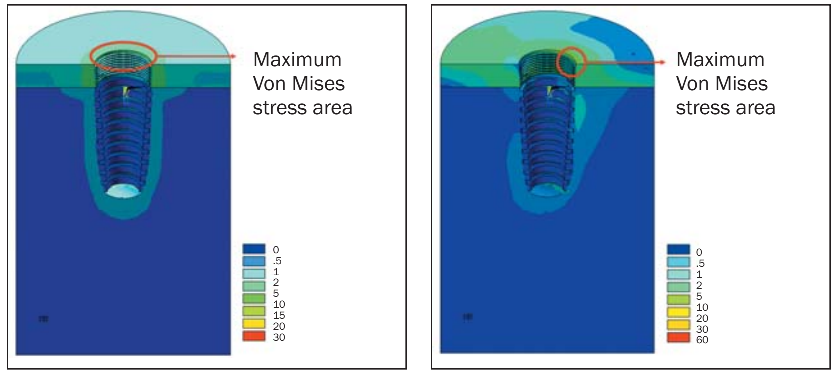
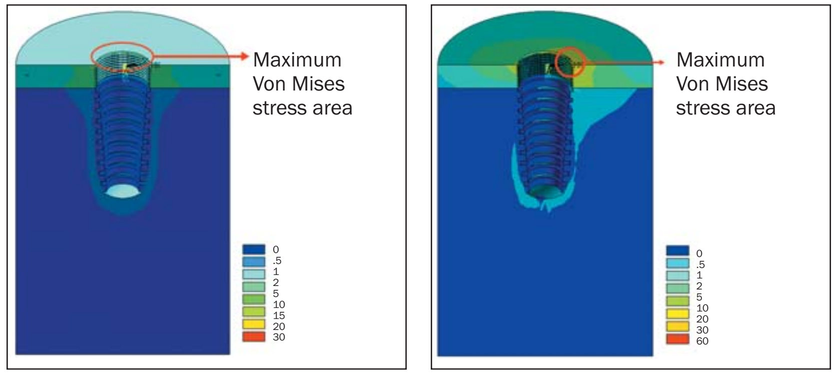


Fig 6a and 6b Von Mises stress distribution in bone surrounding an osseointegrated 1-piece implant with different loading conditions: *(a)* the vertical loading condition; *(b)* inclined loading condition (30 degrees). Stress is shown in MPa.



Figs 7a and 7b Von Mises stress distribution in bone surrounding an osseointegrated internal-hex implant with different loading conditions: *(a)* the vertical loading condition; *(b)* inclined loading condition (30 degrees). Stress is shown in MPa.



Figs 8a and 8b Von Mises stress distribution in bone surrounding osseointegrated external-hex implant with different loading conditions: *(a)* the vertical loading condition; *(b)* inclined loading condition (30 degrees). Stress is shown in MPa.

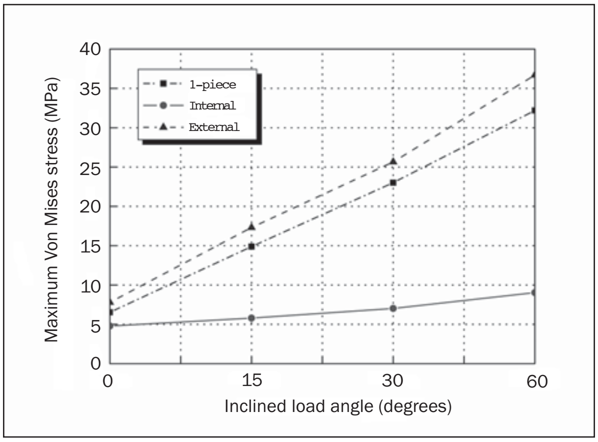


Fig 9a Maximum Von Mises stresses of different implant systems in bone as functions of inclined load angle.

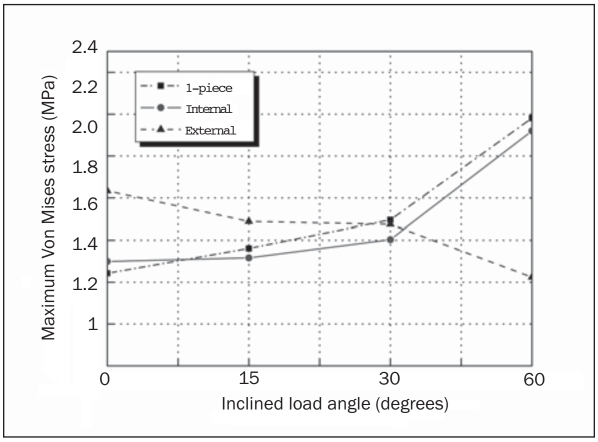
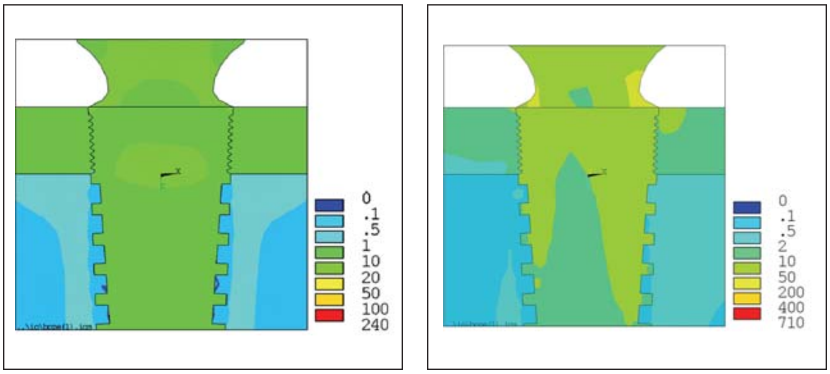
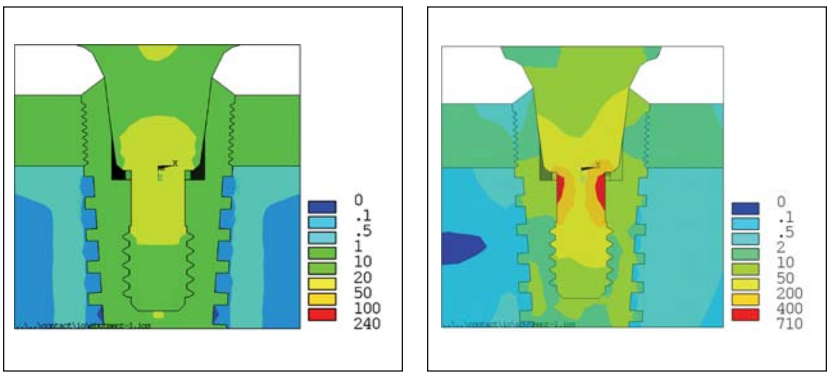


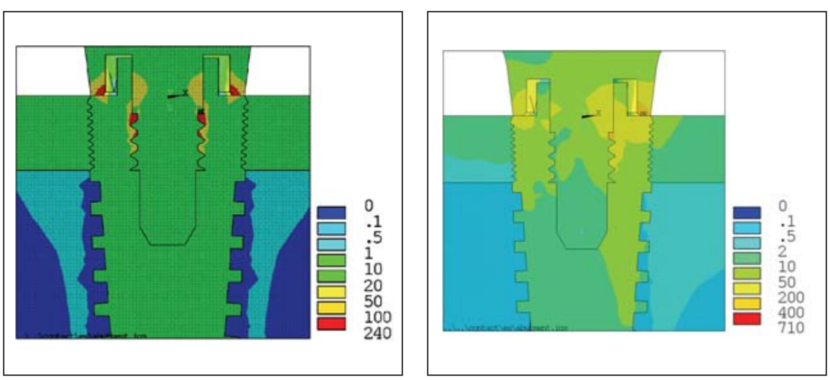
Fig 9b Maximum Von Mises stresses of different implant systems in spongious bone as functions of inclined load angle.



Figs 10a and 10b Von Mises stress distributions in the 1-piece implant system for *(a)* the vertical loading condition and *(b)* the inclined loading condition (30 degrees).



Figs 10c and 10d Von Mises stress distributions in the internal-hex implant system for *(c)* the vertical loading condition and *(d)* the inclined loading condition (30 degrees).

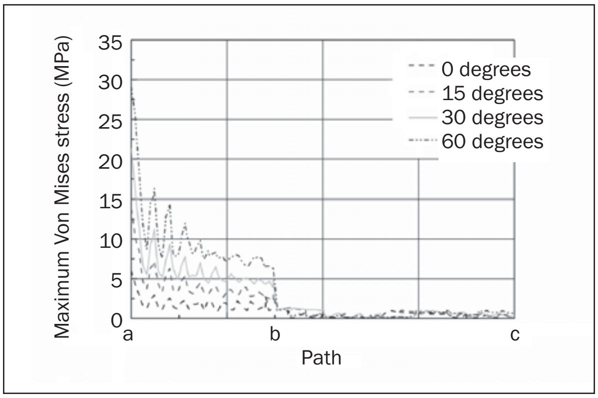


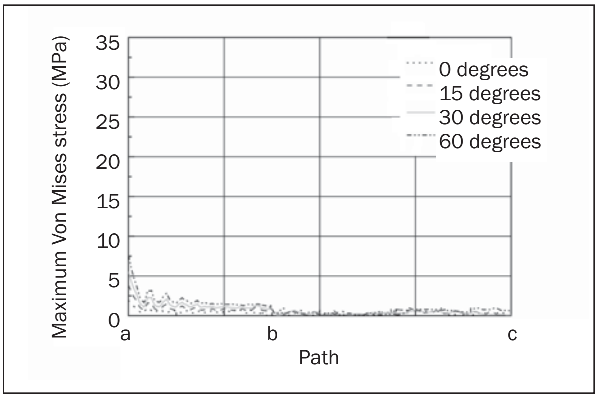
Figs 10e and 10f Von Mises stress distributions in the external-hex implant system for *(e)* the vertical loading condition and *(f)* the inclined loading condition (30 degrees).

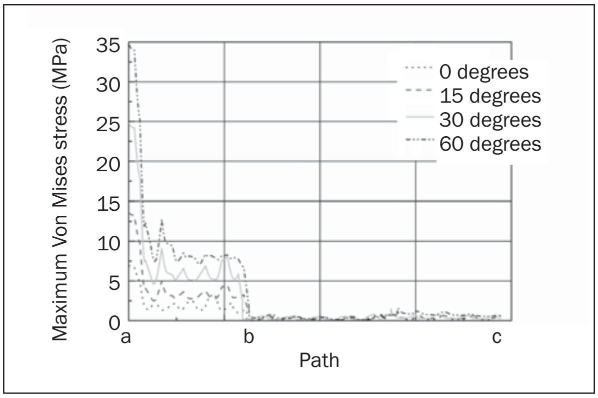
implant and spongious bone. The external-hex implant generated the largest maximum Von Mises stress, whereas the 1-piece implant generated the smallest maximum Von Mises stress in spongious bone under vertical loading conditions. The maximum Von Mises stresses generated in spongious bone by the 1-piece implant and the internal-hex implant increased as the angle of inclined load increased. However, only small differences in the maximum Von Mises stresses in spongious bone between the 1-piece implant and the internal-hex implant were observed with respect to the variation of inclined load angle. The maximum Von Mises stress in spongious bone by the external-hex implant decreased as the angle of inclined load increased; this trend was opposite of those of the other implant systems.

From these results, it is reasonable to conceive that most of the applied load transmitted to the external-hex implant was taken at the compact bone adjacent to the first microthread of the implant, such that a relatively small amount of load was taken at the interface between the bottom of the implant and spongious bone because of the presence of a butt joint configuration in the case of the external-hex implant as the angle of inclined load increased. It is believed that these differences in maximum stresses were caused by a difference in the load transfer mechanism for various abutments.

Figure 10 shows the stress distribution generated in 3 different types of implant systems. The different loading transfer mechanisms for vertical and inclined loading conditions are demonstrated. In the case of 1-piece implants, as the abutment and implant are







Figs 11a to 11c Changes of Von Mises stresses from points a to b (compact bone) and from points b to c (spongious bone) (see Fig 5) for a 1-piece implant *(above)*, an internal-hex implant *(above right)*, and an external-hex implant *(right)*.

joined as a single piece, the load is transferred more evenly, not only in the implant, but also in bone. However, the maximum stress generated in the compact bone with the 1-piece implant was always higher than that found with the internal-hex implant, regardless of the inclined load angle. The location of maximum Von Mises stress generated in the 1-piece implant at the boundary between the abutment and implant was such that the top part of the compact bone was where the highest Von Mises stress was generated. In the case of the internal-hex implant, since the location for the maximum Von Mises stress developed was at the neck of the abutment screw, indicating that the region for the maximum load transfer was away from the top part of compact bone, where the highest Von Mises stress was generated, the maximum Von Mises stress was the lowest among those generated in compact bone. The contact condition with friction between abutment and implant in the tapered joints and at the abutment neck reduced the effect of bending caused by the horizontal component of inclined load. Also, the sliding effect caused by the contact condition reduced the stress concentration at the top of the compact bone region.

In the case of the external-hex implant, the maximum Von Mises stress was the highest among those generated in the compact bone for all loading conditions because, like the 1-piece implant, the location of maximum Von Mises stress generated in the external-hex implant at the boundary between the abutment and the implant was adjacent to the top part of the compact bone, where the highest Von Mises stress was generated. However, the level of the maximum Von Mises stress developed at the boundary between the abutment and implant in the external-hex implant was higher than that generated in the 1-piece implant, since the load transfer area between abutment and implant was reduced in the case of external-hex implant. In the case of the 1-piece implant, the stress was well distributed near the boundary of the abutment and implant since the abutment and implant were connected as 1 piece. However, in the case of the external-hex implant, the stress was concentrated at the interface between the abutment and implant adjacent to a butt joint because a smaller allowable contact area was provided between abutment and implant than that of the 1-piece implant.

Figure 11 shows the changes in Von Mises stresses at the interface between implant and bone along the length of the implant (Fig 5) and in the bone surrounding the 3 implant types for vertical and inclined loads. As shown in the figure, the maximum Von Mises stresses were generated at the top of the compact bone region and decreased gradually toward the bottom of implant. Significant reduction in Von Mises stresses for all implant systems was observed at the boundary between compact and spongious bone because of the low elastic modulus of spongious bone.

DISCUSSION

To achieve stable osseointegration for implant restoration, the generation of high stress concentration or distribution in bone should be avoided, since the high level of stress concentration or distribution can induce severe resorption in the surrounding bone, leading to gradual loosening and, finally, complete loss of the implant. Therefore, study of the effect of abutment type on stress distribution in bone is important.

The effect of abutment type on stress distribution in bone under vertical and inclined loads was investigated by performing finite element analysis with contact friction at the interfaces between the abutments and implants for 3 types of implant systems. One-piece, internal-hex, and external-hex implant systems were selected for the study. The maximum Von Mises stress occurred at the region in compact bone adjacent to the first implant microthread of the implant for all implant systems with different abutments for both vertical and inclined loading conditions.

For the vertical loading condition, the stress distribution in bone for all 3 implant systems showed similar trends, but the maximum Von Mises stresses generated in bone were all different for the implant systems. For the inclined loading condition, the stress distribution in bone for all 3 implant systems showed noticeable differences. These differences for vertical and inclined loading conditions were caused by the change in load transfer mechanism related to the different abutment types. Also, the size of the contact area between the abutment and implant significantly influenced the stress distribution and magnitude of maximum Von Mises stress generated in bone for all implant systems.

The maximum Von Mises stress increased as the inclination angle of applied load increased for each implant system. The lowest maximum Von Mises stress was obtained in bone surrounding the internal-hex implant system, and the highest maximum Von Mises stress was obtained in bone surrounding the external-hex implant system for all loading conditions. Differences in the maximum Von Mises stress increased with increase of angle of inclined load. The internal-hex implant system generated the lowest maximum Von Mises stresses for all loading conditions because of the reduction in effect of bending caused by the horizontal component of inclined load by sliding in the tapered joints between the implant and abutment.

When the variation in Von Mises stresses at the interface between different implant systems and bone was monitored for vertical and inclined loads, the Von Mises stresses decreased gradually from the marginal bone level to the apex of the implant. Significant reduction in Von Mises stress was observed at the boundary between compact and spongious bones because of the relatively low elastic modulus of spongious bone.

ACKNOWLEDGMENT

This study was assisted by Korea Science and Engineering Foundation (Basic research grant R01-2002-000-00289-0).

REFERENCES

1. Rieger MR, Adams WK, Kinzel GL, Brose MO. Alternative materials for three endosseous implants. J Prosthet Dent 1989;61: 717–722.
2. Rieger MR, Adams WK, Kinzel GL. A finite element survey of eleven endosseous implants. J Prosthet Dent 1990;63:457–465.
3. Clelland NL, Ismail YH, Zaki HS, Pipko D. Three-dimensional finite element stress analysis in and around the Screw-Vent implant. Int J Oral Maxillofac Implants 1991;6:391–398.
4. Siegele D. Numerical investigations of the influence of implant shape on stress distribution in the jaw bone. Int J Oral Maxillofac Implants 1989;4:333–340.
5. Holmgren EP, Seckinger RJ, Kilgren LM, Mante F. Evaluating parameters of osseointergated dental implant 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.
6. Chun HJ, Cheong SY, Han JH, Heo SJ, Chung JP, Rhyu IC. Evaluation of design parameters of osseointegrated dental implants using finite element analysis. J Oral Rehabil 2002;29:565–574.
7. Pierrisnard L, Hure G, Barquins M, Chappard D. Two dental implants designed for immediate loading: A finite element analysis. Int J Oral Maxillofac Implants 2002;17:353–362.
8. Bozkaya D, Muftu S. Mechanics of the tapered interference fit in dental implants. J Biomech 2003;36:1649–1658.
9. Hansson S. A conical implant-abutment interface at the level of the marginal bone improves the distribution of stresses in the supporting bone. Clin Oral Implants Res 2003;14:286–293.
10. Merz BR, Hunenbart S. Mechanics of the implant–abutment connection: An 8-degree taper compared to a butt joint connection. Int J Oral Maxillofac Implants 2000;15:519–526.