Abstract

Background and aims. Finite elemental analysis is an efficient technique for investigating biomechanical interactions of different implant designs. The purpose of this study was to investigate the effect of cylindrical and tapered implants with different degree of tapering and similar lengths on the stress and strain distribution in the bone and implant.

Materials and methods. One cylindrical and five types of tapered implants with degrees of tapering from 0.02 to 0.16 were modeled to this study. The implant material was grade IV titanium and abutment was grade ELI titanium. The bone model used comprised of compact and spongious bone assumed to be homogeneous, isotropic and linearly elastic.

Results. With increased degree of implant tapering, the von Mises stress and strain increased in the bone. However, at the neck of implant, the most sensitive area, with increase in degree of tapering, both stress concentration and strain decreased. The lowest stress and strain were generated in the most tapered implant.

Conclusion. Based on the results, cylindrical screw implant generated the lowest maximum von Mises stress in cortical bone and tapered implant type 5 with highest taper degree generated the highest maximum von Mises stress.

Key words: Biomechanics, dental implants, finite element analysis, stress distribution.

Introduction

Treatment planning for oral rehabilitation is nowadays mostly based on using dental implants, the function of which are transferring loads to the surrounding biological tissue. Although long-term clinical studies report 95% survival for mandibular and 65% to 85% survival for maxillary implants, failure may result from loss of osseointegration or component failure subsequent to restoration and may be related to unfavorable loading or to high stress concentrations. There are more than 90 dental implant body designs available. Biomechanical rationale of dental implant designs may evaluate these designs as to their efficacy to many biomechanical loads. Review of literature shows different implant survival rates and different...
marginal bone loss for various implant body designs.\textsuperscript{3-13}

Dental implants have macroscopic and microscopic design components. Variation in component design among implant systems may lead to different stress/strain distribution, thus altering the transmission of forces to surrounding bone.\textsuperscript{14} Smooth-sided, cylindrical and tapered implants have a component of compressive load to be delivered to the implant-bone interface, depending on the degree of taper.\textsuperscript{15} However, there are limitations for the amount of implant tapering.\textsuperscript{2}

Finite elemental analysis (FEA) is an efficient technique for investigating biomechanical interactions of different implant designs. Numerous studies have been performed to assess force distribution following load application to implants of various dimensions.\textsuperscript{4,5,16-18}

To enhance clinical success, it is necessary to understand how the stress concentration on implants is affected by the shape, width, and height of thread. The use of the finite element method (FEM) in implant biomechanics analysis offers many advantages over other methods in simulating the complexity of clinical situations.\textsuperscript{15}

The aim of this study was to determine the effect of different tapered implants (with different gradient in bone and similar vertical length) and cylindrical screw on the stress and strain distribution in bone and implant.

Materials and Methods

Implants and abutments were designed and then analyzed with Ansys version 10 computer software (Swanson, Washington, USA). Because the models were symmetric to a plane as well as axis-symmetric (as shown in Figure 1), we designed two-dimensional models.

Compact and spongious bone were assumed to be homogeneous, isotropic and linearly elastic. The thickness of the compact bone layer in the model was assumed to be 1.3 mm as shown in Figure 2. The overall dimensions of the bone were 23.4 mm in height, 23.8 mm in mesiodistal length and 9 mm in buccolingual width. The material properties used in the finite element analysis are shown in Table 1.

Five tapered dental implants with tapering degree of 0.02, 0.06, 0.1, 0.12, and 0.16; and one cylindrical implant with similar length (13.6 mm) and diameter (4 mm) were used; the abutment height was 6 mm. Implant pitch for all implants used is shown in Figure 3. No thread was modeled at the cortical bone level.

All materials used in the models were considered to be, homogenous, isotropic and linearly elastic. The implant material was titanium grade IV and abutment was titanium grade ELI. For the titanium implant and titanium abutment, elastic module of 114 GPa and 113.8 GPa, and for compact bone and spongious bone, elastic module of 14 Gpa and 1.5 Gpa were used.

Meshing and loading

Because of its symmetry, only half of the model was meshed with plane 83 elements, and 0.02 mm distance between implant and abutment was meshed by contact element using contact manager. This is the most important stage of meshing. Obviously, use of different elements and also distinct condition at the interface of

Table 1. The material properties used in the finite element analysis

<table>
<thead>
<tr>
<th>Material</th>
<th>Elastic module (GPa)</th>
<th>Poison ratio</th>
</tr>
</thead>
<tbody>
<tr>
<td>Titanium grade ELI (abutment)</td>
<td>114</td>
<td>.38</td>
</tr>
<tr>
<td>Titanium grade IV (implant)</td>
<td>113.8</td>
<td>.34</td>
</tr>
<tr>
<td>Cortical bone</td>
<td>14</td>
<td>.3</td>
</tr>
<tr>
<td>Spongious bone</td>
<td>1.5</td>
<td>.3</td>
</tr>
</tbody>
</table>
implant and abutment causes different results in stress and strain.

Meshing consisted of 2861 elements and 8299 nodes. The two-dimensional models were constrained in all directions at the nodes on the upper edge and in direction on the right boundary. Important to the symmetry in the model is boundary condition (BC) symmetry on left boundary. Finally, a pressure of 30 psi, which is close to normal occlusal pressure, was applied axially on lower edge of the abutment.

**Results**

**Stress/strain distribution in bone**

As shown in Figures 6 and 7, in all types of implants assessed, von Mises stress distribution in the bone was similar; however, maximum von Mises stress increased as implants were more tapered. In all types of implants, maximum von Mises stress occurred in a region of cortical bone adjacent to implant neck and lower bound of cortical bone. The highest maximum stress was generated in tapered implant type 5 (0.572 MPa, Figure 7b).

Maximum von Mises strain was generated in the spongious bone. Strain distributions in all types of implant in spongious bone were similar, with a slight increase as implants were more tapered. Maximum strain was seen in the apex of implant in the spongious bone. With only 30 psi auxiliary pressure, Maximum von Mises strain was observed in type 5 tapered implant (strain $0.11 \times 10^{-3}$, Figure 9b). Results of stress and strain distribution are shown in Table 2 and maximum von Mises stress and strain in bone are plotted in Figures 10a and 10b.

**Table 2. Maximum von Mises stress and strain in bone for all types of implant under 30 psi auxiliary pressure**

<table>
<thead>
<tr>
<th>Implant type</th>
<th>Maximum von Mises stress (MPa)</th>
<th>Maximum von Mises strain ($\times 10^{-3}$)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Cylindrical</td>
<td>.466</td>
<td>.881</td>
</tr>
<tr>
<td>Conic 1</td>
<td>.475</td>
<td>.850</td>
</tr>
<tr>
<td>Conic 2</td>
<td>.500</td>
<td>.941</td>
</tr>
<tr>
<td>Conic 3</td>
<td>.523</td>
<td>.981</td>
</tr>
<tr>
<td>Conic 4</td>
<td>.538</td>
<td>1</td>
</tr>
<tr>
<td>Conic 5</td>
<td>.572</td>
<td>1.1</td>
</tr>
</tbody>
</table>
Figure 6. von Mises stress distribution in the cylindrical type from 3431 Pa to 466118 Pa (a) and conic type 1 with 0.2 gradient from 4040 Pa to 475569 Pa (b).

Figure 7. von Mises stress distribution in the conic type 4 with 0.12 gradient from 4865 Pa to 538315 Pa (a) and conic type 5 with 0.16 gradient from 4143 Pa to 572560 Pa; highest max stress is generated in this type (b).

Figure 8. von Mises strain distribution in the cylindrical from $0.926 \times 10^{-6}$ to $0.881 \times 10^{-4}$ Pa (a) and conic type 2 with 0.06 gradient from $0.903 \times 10^{-6}$ to $941 \times 10^{-4}$ Pa (b).
Figure 9. von Mises strain distribution in the conic type 4 with 0.12 gradient from $4.57 \times 10^{-6}$ to $1.01 \times 10^{-3}$ Pa (a) and conic type 5 with 0.16 gradient from $4.42 \times 10^{-6}$ to $1.10 \times 10^{-4}$ Pa; highest max strain is generated in this type (b).

Figure 10. Maximum von Mises stress (a) and strain (b) in all bones under 30 psi axial pressure.

Figure 11. von Mises stress distribution in the cylindrical type from 804 to $1.75 \times 10^7$ Pa (a) and conic type 2 with 0.06 gradient from 345 to $1.46 \times 10^7$ Pa; lowest maximum stress is generated in this type (b).
Stress/strain distribution in implant

As shown in Figures 11 and 12, in all types of implant, maximum stress was generated at the abutment-implant interface. The lowest maximum stress was generated in tapered implant type 2 (Figure 11b), and the highest maximum stress was generated in type 4 tapered implant (Figure 12b). It is, however, important to note that from type 1 to type 4, stress concentration decreased at the neck of implant in the region between cortical bone and abutment. At this region, in implant type 1, stress was 0.38 to 0.58 MPa; however, in type 4, stress at the same region was 0.25 to 0.5 MPa. For all implants, stress distribution gradually increased from neck to apex; however, in tapered implant type 4, stress concentration decreased intensely at the neck, which seems like an area of restricted stress distribution.

A different pattern was observed with strain, as the only similarity of all types of implants was in that max strain was generated at the abutment-implant interface. The lowest maximum strain was generated in tapered implant type 2 (0.129 × 10^4, Figure 13b) and the highest was generated in tapered implant type 4 (0.202 × 10^4, Figure 14b). Table 3 summarizes the results of maximum stress and strain for all implant types evaluated. Charts of maximum von Mises stress and strain are plotted in Figures 15a & 15b.

Discussion

Although implant failures are still unavoidable, the successful use of dental implants has been well docu-
Stress and Strain of Tapered Implants

The mechanisms responsible for implant failure are not fully understood, owing to complications from many related factors, such as loading condition, prosthesis type, implant design, implant position, bone type, and material properties of the bone-implant interface. Bone quality is also an important factor, with more failures found in low quality of bone. These factors are difficult to investigate clinically, because of limited information as well as sample variation.

Dental implant failure may result from loss of osseointegration or component failure subsequent to restoration and may be related to unfavorable loading or to high stress concentrations. Bone quality is also an important factor, with more failures found in low quality of bone. These factors are difficult to investigate clinically, because of limited information as well as sample variation.

Variations in design among dental implant systems may lead to different stress/strain distributions, thus altering the transmission of forces to surrounding bone. In the case of osseointegrated dental implants, occlusal loads are transmitted directly to the surrounding bone. However, in natural teeth, the periodontal ligament acts as an intermediate cushion element to buffer occlusal forces.

Several studies have attempted to minimize crestal bone resorption by increasing the bone-implant contact and, therefore, decreasing stress at the alveolar crest. To enhance clinical success, it is necessary to understand how the stress concentration on implants is affected by the shape, width, and height of thread. The use of the finite element method in implant biomechanics analysis offers many advantages over other methods in simulating the complexity of clinical situations.

Pierrisnard et al performed a finite element analysis of 3.75-mm-wide hex-headed screw-type implants.
to 12 mm in length and reported that the magnitude and distribution of stress to the bone was constant and independent of implant length. These findings were contradicted by Petrie & Williams, who performed a finite element analysis of implants with diameters of 3.5 to 6 mm and lengths of 5.75 to 23.5 mm, placed in molar regions, and reported a reduction in peak crestal stress following force application with implants of increasing diameter and/or length. In contrast, Holmgren et al and Himmlova et al demonstrated that force application resulted in greatest force concentration at the bone crest and that implant length had no effect on either the magnitude of peak stress or stress distribution to the supporting bone. The preponderance of finite element analyses demonstrates that implant length has no effect on the magnitude of stress experienced by the supporting alveolar bone crest around implants, which would seem to support the use of shorter implants if they offer specific advantages in given clinical situations.

The generation of high stress distribution or concentration in the bone should be avoided to achieve stable osteointegration for implant restoration; therefore, the influence of implant on stress and strain distribution in the bone must be investigated. In this study, the effect of implant on stress distribution in the bone under vertical pressure was investigated by performing finite element analysis using Ansys computer software with contact friction at the interface between the abutment and implants. Six types of implant, one cylindrical and five tapered, with similar vertical lengths and different degrees of tapering were selected. Maximum von Mises stress in the bone occurred at the region of cortical bone adjacent to implant neck and lower bound of cortical bone.

In the present study, maximum von Mises stress occurred at the implant-abutment interface for all implants. Maximum stress was registered at the crestal bone, which is the most sensitive area. With increase in the degree of tapering, stress concentration decreased. In cortical bone as the degree of tapering increased, the maximum von Mises stress and strain also increased; therefore, the maximum von Mises is generated in tapered implant type 5 with 0.16 tapering. The highest maximum von Mises stress and strain was generated in tapered implant type 4 with 0.12 degree tapering and the lowest maximum von Mises stress and strain was seen in tapered implant type 2 with 0.06 degree tapering.

**Conclusion**

Based on the results of the analyses, it was found that cylindrical screw implant generated the lowest maximum von Mises stress in cortical bone and tapered implant type 5 with highest tapering generated the highest maximum von Mises stress, probably because of reduced and sharper surface area.

**References**


