Title
The influence of prosthesis designs and loading conditions on the stress distribution of tooth-implant supported prostheses

Author(s)
Zhiyong, L; Arataki, T; Shimamura, I; Kishi, M

Journal
Bulletin of Tokyo Dental College, 45(4): 213-221

URL
http://hdl.handle.net/10130/275
INTRODUCTION

Based on numerous long-term clinical and theoretical studies, it has been concluded that freestanding fixed partial prostheses should be the first choice whenever possible. However, anatomical limitations of space for implants or failure of an implant to osseointegrate may create a situation in which it would be desirable to connect the implants to the teeth. After the tooth-implant supported prosthesis (TISP) was first described by Ericsson et al. in 1986, many clinicians were against this particular therapy because of the differing mobilities...
of implants and teeth. Now, after several clinical long-term studies, it has been concluded that TISP is an equally predictable treatment as the implant supported prosthesis with respect to implant survival rate and loss of marginal bone.

Researchers and clinicians have discussed the theoretical risk inherent in connecting an immobile osseointegrated implant to a movable tooth. A tooth with a sound periodontal ligament have mobility characteristics of between 50 and 200 \( \mu \text{m} \), while osseointegrated implants demonstrate a mobility of less than 10 \( \mu \text{m} \). It has been suggested that the physiologic movement of the tooth causes the prosthesis to act as a cantilever, resulting in implant overload. A potential consequence of such overloading may be peri-implant marginal bone resorption, which may eventually cause failure of the osseointegration. Another problem associated with TISP is the intrusion of the tooth. A survey has shown that the incidence of intrusion of tooth was 3.5%, although the cause remains unknown.

Many stress analysis studies have revealed that high stress concentrates around the implant neck where bone resorption is most often seen. To reduce and redistribute the stress concentration away from the implant neck, many investigators have advocated various methods such as applying stress absorbing elements or non-rigid connectors. Studies have demonstrated that stress absorbing elements may be effective only when their resiliency is about the same order or magnitude as the periodontal ligament; it is difficult to meet this demand. A non-rigid connector has the ability to separate the splinted units, thus compensating for the different degrees of mobility between the implant and tooth. Lin et al. analyzed the stress distribution of three units TISP using rigid and non-rigid connectors and found that the stress-breaking function of the keyway device becomes obvious only when the splinting system receives smaller occlusal forces. Moreover, the clinical studies indicate that, if the implant has to connect with the teeth, the connection should be fully rigid to prevent abutment tooth intrusion as long as the etiology of teeth intrusion remains unanswered.

The finite element method (FEM) is widely used to analyze the stress distribution. Previous FEM studies have focused on the stress distribution when one tooth and one implant were connected rigidly or non-rigidly, but there are other clinical situations such as those in which two teeth are connected with one implant or one tooth is connected with two implants. The purpose of the present study was to investigate the influence of prosthesis designs and loading conditions on the stress distributions in the TISP.

**MATERIALS AND METHODS**

1. **2D Finite element models**
   1) The basic models
   The basic model of single tooth and single
Implant (Fig. 1) was computed and tested. The alveolar bone, including cortical bone and spongy bone, was 1.7 mm and 1.8 mm thick, respectively. The model of single tooth simulated the premolar. The root was 13.0 mm, and the crown was 8.0 mm. The periodontal ligament was 0.2 mm in width. A layer of cortical bone, 0.4 mm in thickness, was added between periodontal ligament and spongy bone. The model of single implant simulated a Brånemark implant 10.0 mm in length and 3.75 mm in diameter. The abutment was 4.0 mm in height and 4.5 mm in diameter. According to a previous study, a layer of material 0.5 mm in thickness was inserted between the abutment and the fixture to simulate the condition of the abutment screwed on the fixture. Between the cortical bone and the fixture, a layer of material 0.2 μm in thickness was placed to simulate the situation of osseointegration. The material properties are listed in Table 1.

The validity of the basic model was tested by comparing its displacement with actual measurements under the same testing loading. The test loading was 10 N vertically and horizontally for the tooth and 20 N horizontally and 50 N vertically for implant, respectively. From the testing results listed in Table 2, the validity of the basic models was confirmed.

2) Experimental models

Experimental models were composed from the basic models (Fig. 2). The four experimental models simulated four different prosthesis designs. Model TPF represented one pontic between one tooth and one implant.
as abutment, TPPF represented two pontics between one tooth and one implant as abutment, TTPF represented one pontic with two teeth and one implant as abutment, and TPFF represented one pontic with one tooth and two implants as abutment. The fixture was placed distally of the tooth. The superstructure of all the models was made of gold alloy. The pontic was 7.5 mm in length. The fixed points of the models were the nodal point of the mesio-distal cortical bone and the total nodal point of the inferior marginal of the model. The thickness of all the models was 1.5 mm.

3) Reference models

The model of the two-unit-cantilever bridge was computed as the reference model (Fig. 3). The model TP and FP represented that one tooth and one fixture was used as abutment. The reference models represented the worst clinical situation, the stresses in reference models were considered as 100%.

2. Loading conditions

Six loading conditions were applied to each model. L1 was vertical loading only on the tooth; L2, vertical loading only on the implant; L3, vertical loading only on the pontic; L4, vertical loading on the tooth and pontic; L5, vertical loading on the implant and pontic; L6, vertical loading on the whole prosthesis. The loading was 50N for each node on the middle of the occlusal surface.

3. Analyzing method

The analysis of the finite element model was performed by personal computer (Dell Inspiron 600m, Dell Inc., U.S.A) using the linear static analysis module of the general purpose finite element analysis program (COSMOS/M ver.2.85, Structural Research and Analysis Co., U.S.A.).

RESULTS

1. Von Mises stresses of experimental models

The Von Mises stresses of experimental models were listed in the Tables 3, 4, 5. The highest stress for the tooth was 1.49 MPa in
Table 3 Von Mises values of models TPF and TPPF (MPa)

<table>
<thead>
<tr>
<th>Model</th>
<th>Loading conditions</th>
<th>Tooth</th>
<th>Fixture</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td>Mesial</td>
<td>Apical</td>
</tr>
<tr>
<td>TPF</td>
<td>L1</td>
<td>0.82</td>
<td>0.13</td>
</tr>
<tr>
<td></td>
<td>L2</td>
<td>0.01</td>
<td>0.09</td>
</tr>
<tr>
<td></td>
<td>L3</td>
<td>0.27</td>
<td>0.03</td>
</tr>
<tr>
<td></td>
<td>L4</td>
<td>1.08</td>
<td>0.16</td>
</tr>
<tr>
<td></td>
<td>L5</td>
<td>0.29</td>
<td>0.09</td>
</tr>
<tr>
<td></td>
<td>L6</td>
<td>1.10</td>
<td>0.10</td>
</tr>
<tr>
<td>TPPF</td>
<td>L1</td>
<td>0.91</td>
<td>0.15</td>
</tr>
<tr>
<td></td>
<td>L2</td>
<td>0.003</td>
<td>0.06</td>
</tr>
<tr>
<td></td>
<td>L3</td>
<td>0.59</td>
<td>0.08</td>
</tr>
<tr>
<td></td>
<td>L4</td>
<td>1.49</td>
<td>0.22</td>
</tr>
<tr>
<td></td>
<td>L5</td>
<td>0.59</td>
<td>0.08</td>
</tr>
<tr>
<td></td>
<td>L6</td>
<td>1.49</td>
<td>0.18</td>
</tr>
</tbody>
</table>

Table 4 Von Mises values of model TTPF (MPa)

<table>
<thead>
<tr>
<th>Model</th>
<th>Loading conditions</th>
<th>Mesial tooth</th>
<th>Distal tooth</th>
<th>Fixture</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td>Mesial</td>
<td>Apical</td>
<td>Distal</td>
</tr>
<tr>
<td>TTPF</td>
<td>L1</td>
<td>1.03</td>
<td>0.16</td>
<td>0.96</td>
</tr>
<tr>
<td></td>
<td>L2</td>
<td>0.003</td>
<td>0.04</td>
<td>0.005</td>
</tr>
<tr>
<td></td>
<td>L3</td>
<td>0.09</td>
<td>0.10</td>
<td>0.14</td>
</tr>
<tr>
<td></td>
<td>L4</td>
<td>1.10</td>
<td>0.12</td>
<td>1.09</td>
</tr>
<tr>
<td></td>
<td>L5</td>
<td>0.09</td>
<td>0.14</td>
<td>0.15</td>
</tr>
<tr>
<td></td>
<td>L6</td>
<td>1.10</td>
<td>0.12</td>
<td>1.10</td>
</tr>
</tbody>
</table>

Table 5 Von Mises values of model TPFF (MPa)

<table>
<thead>
<tr>
<th>Model</th>
<th>Loading conditions</th>
<th>Tooth</th>
<th>Mesial fixture</th>
<th>Distal fixture</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td>Mesial</td>
<td>Apical</td>
<td>Distal</td>
</tr>
<tr>
<td>TPFF</td>
<td>L1</td>
<td>0.73</td>
<td>0.07</td>
<td>0.35</td>
</tr>
<tr>
<td></td>
<td>L2</td>
<td>0.02</td>
<td>0.15</td>
<td>0.05</td>
</tr>
<tr>
<td></td>
<td>L3</td>
<td>0.24</td>
<td>0.04</td>
<td>0.27</td>
</tr>
<tr>
<td></td>
<td>L4</td>
<td>0.97</td>
<td>0.08</td>
<td>0.61</td>
</tr>
<tr>
<td></td>
<td>L5</td>
<td>0.26</td>
<td>0.18</td>
<td>0.32</td>
</tr>
<tr>
<td></td>
<td>L6</td>
<td>0.98</td>
<td>0.15</td>
<td>0.66</td>
</tr>
</tbody>
</table>
the model TPPF under L4 and L6, the lowest was 0.003 MPa in the model TTPF under L2. The highest stress for the fixture was 4.86 MPa in the model TPPF under L6, the lowest was 0.04 MPa in the model TPFF under L3.

2. Relatives stresses of experimental models

The relative stresses were shown in the Fig. 4. The lowest stress was found in the model TPFF for both tooth and fixture, there was no difference between model TPF and TTPF.

3. The stresses under different loading conditions

The stresses under different loading conditions were illustrated in the Fig. 5. The highest stress for fixture was found in L4 and the lowest in L2, the highest stress for tooth was found in L6 and lowest in L2. The stresses under L4 and L6 were almost the same.
4. The rotation stress of the tooth

The rotation stress of the tooth was illustrated in the Fig. 6. The highest stress was found in the model TTPF and the lowest in the model TPFF.

DISCUSSIONS

1. About the analysis method

The finite element method is one of the most frequently used methods for stress analysis in both industry and science. It is used for analyzing hip joints, knee prostheses, and dental implants. The results of the FEA computation depend on many individual factors, including material properties, boundary conditions, interface definitions, and also on the overall approach to the model. Because the finite element models are mathematical models of the real objection or phenomenon, it is usually impossible to reproduce all the details of natural behavior. On the other hand, the mathematical models have enormous advantages over in vivo testing, precisely because the mathematical models are virtual; they exist in the computer and are completely controllable. The researcher can easily change the test conditions, the model parameters, and the geometry, can simulate any desirable response, and can repeat the test at any time. Obviously, all of these things are impossible during in vitro tests. Therefore, the tested and verified mathematical models provide the researchers with a very powerful tool for analysis.

The models used in present study were computed based on the basic model. The basic model was tested to precisely simulate real physical conditions. The material properties were adjusted according to the study of Takasaki et al. The most difficult thing was to define the material properties of periodontal ligament (PDL). The mechanical behavior of the PDL changes nonlinearly, depending on the magnitude and duration of the load applied. Yoshida et al. and Provatis pointed out that Young’s modulus of elasticity of the PDL increased almost exponentially with the load increment. The value was found to be approximately 0.68 MPa under a load of 1N. In the present study, the PDL was assumed to be linear elastic, homogeneous, and isotropic as the result of numeric convergence considerations and a larger variation for the PDL physical properties in the literature. The Young’s modulus of PDL was set to 1–24 MPa and increased from 1 MPa from apical to 24 MPa at cervical cortical bone. The displacement of the basic model was in the range of actual measurement. Therefore, we
believe that the models used in present study well simulated in vivo conditions.

2. The influence of prosthesis design on the stress distribution

In the present study, the influence of prosthesis designs on the stress distribution in TISP was investigated by comparing the stresses in experimental models with those of reference models. The lowest stress was found in the model TPFF for both tooth and implant; there were no differences between the TPF and TTPF models. The results indicate that there is no advantage in connecting more teeth as abutments in TISP, but, if there are more fixtures, the stress on the tooth decreases. The findings supported the opinion that the teeth do not stabilize implants; on the contrary, fully integrated implants can stabilize periodontally compromised teeth.

3. The influence of loading conditions on the stress distribution

Studies have proven that the loading conditions influence the stress distributions. It has been recommended to minimize the occlusal loading force on the pontic area through occlusal adjustment procedures in order to redistribute stress within the implant system in the TISP. In this study, the lowest stress was found when vertical loading alone was applied to the implant. For the implant, the highest stress was found in L4 and the lowest in L2. For the tooth, the highest stress was found in L6 and lowest in L2. The stress around the tooth between L4 and L6 was almost the same, so we concluded that the worst loading condition for TISP was loading on the tooth and the pontic; the best was loading on the implant.

4. The rotation stresses of tooth under vertical loading

Lin et al. found that the displacement of the tooth was 11 times that of implant under vertical loading in three-unit-fixed bridge with one tooth and one fixture as abutment connected with a rigid connector. Because the superstructure was connected rigidly, the difference in displacement between the tooth and the implant resulted in a stress concentration around the neck of implant and the rotation of tooth. In this study, the highest stress was found at the 3/4 position of the root. The highest stress was found in the model TTPF, and the lowest in the model TPFF. Because the fixture was rigidly integrated with bone, the displacement is smaller than that of the tooth. Under vertical loading, the fixture acts like the pivot in TISP, while the superstructures act as force arms. The stress increases as the lengths of the force arms increase; that’s the reason the highest stress was found at the mesial tooth in the model TTPF. In a traditional fixed bridge using only the tooth or the implant as the abutment, if the supporting ability of the abutment is inadequate, more abutments should be added to the weak side to share the load. In TISP, the fixture is considered stronger than the tooth, but connecting more teeth as abutments does not serve to reduce the stresses.

CONCLUSIONS

Within the limitation of this study based on 2D finite element analysis of TISP, the following conclusions can be drawn:
1. The fixture takes the major portion of the functional loading when it is connected with the tooth.
2. There is no significant influence on reducing stress in TISP as a result of connecting more teeth as part of the abutment.
3. Minimization of loading on the tooth decreases the stress concentration around the neck of the fixture.
4. Rotation of the teeth may be caused by the tooth intrusion in TISP.

REFERENCES


Reprint requests to:
Dr. Tomohiko Arataki
Department of Removable Partial Prosthodontics,
Tokyo Dental College,
1-2-2 Masago, Mihama-ku,
Chiba 261-8502, Japan