Comparative Study on Material Combinations of Various Dental Implant-abutment Systems

Yu-lin ZHOU¹, Zhi-hong MAO¹,*, Jian-sheng WANG¹ and Qiong-yao WANG¹
¹Division of Intelligence Manufacturing, Wuyi University, China
*Corresponding author

Keywords: Materials, Dental implants, Three-dimensional finite element analysis.

Abstract. Objective: The aim of the study was to evaluate the effect of material combinations on stress distribution at implant-bone interface, which benefits the selection of implant-abutment system materials; Methods: The simplified models of mandibular bone, implant and mandibular molar were established by Geomagic studio and Solidworks. To assess the stress distribution of implants and the peri-implant bone tissue under different implant-abutment system material combinations. All were subjected to static loading and analyzed by finite element method. Results: The results showed that the stress distribution of each group was similar. The stress of the implant was concentrated at the upper thread, and the maximum stress appeared at the neck of the implant. The stresses in abutment were highest at the contact region around the implant. The maximum stress of the screw appeared in the first thread. The stresses in cortical bone and trabecular bone were highest at the crestal region around the implant. Conclusions: The use of implant-abutment system components with different material is possible, and optimal implant-abutment system material design need to consider the compatibility with abutment and implant material, so that we can acquire better comprehensive performance.

Introduction

Primary stability of dental implant has been reported as one of the most important factor for bone integration. Several underlying factors have been discussed to affect the stability of the implant-abutment system. Factors that would influence the initial stability include implant-abutment system materials, implant structure parameters, implant-bone interaction site and implant technology[1]. The integration of material was paid attention as to affecting the stability of the implant under biting force. Therefore, material selection hereof becomes important, so certain implant-abutment systems are known to all that they are manufactured with commercial pure titanium, titanium alloy, as well as other metal materials(zirconia, gold alloy, stainless steel, chromium alloy)[2].

According to the literature[3,4], material composition of coating surfaces, the ratio of implant materials and modification in implant-abutment system are all important factors, which affect the stability of implant. However, despite the material change in implant affect initial stability, few things are known regard the effect of dental implant-abutment system material combination on the stress distribution under masticatory forces, and thus the aim of this study was to explore the effects of various implant-abutment system material combination. The stress distribution of implant-abutment system and surrounding bone tissue would be measured in this study.

Material and Methods

Study Implant-abutment Systems

The implant-abutment system includes implant, abutment and screw. The experimental implant has a V-thread pitch of 1.0mm and thread depth of 0.35mm. The length of the implant is 10mm and the diameter is 4.1mm. The abutment used in this study has the maximum diameter of 4.5mm. Each screw is 2mm in diameter with M2*0.25 standard screw.
Material Selection

For the purpose of the study, 6 experimental materials were selected to conduct 6*6*6 orthogonal tests. Some of the existing research materials Zirconia (Z), Titanium-zirconium alloy (TZ), Commercially pure Titanium (CP Ti), Ti-Nb-Zr alloy (TNZ), Gold alloy (G), Ti-24Nb-4Zr-8Sn alloy (Ti2448) were given to implants, abutments and screws respectively. Thus, 216 kinds of material combinations of implant-abutment system have been formed and the models were classified according to implant-abutment-screw materials for express briefly. “CP Ti-Z-TZ” is the symbol indicates that the implant material is Commercially pure Titanium, the abutment material is Zirconia, and the screw material is Titanium-zirconium alloy in this implant-abutment system. The specific system model is shown in Fig. 1. Meanwhile, the implant materials were divided into A, B, C, D, E and F groups. The implant materials in group A were Zirconia, group B were Titanium-zirconium alloy, group C were Commercially pure Titanium, group D were Ti-Nb-Zr alloy, group E were Gold alloy, and group F were Ti-24Nb-4Zr-8Sn alloy, and there are 36 kinds of implant-abutment system material combinations in each group.

![Figure 1. Three-dimensional models of the implant-abutment system.](image)
A, Implant; B, Abutment; C, Screw

Material Properties

All the material’s properties including the implant, abutment, molar, screw, trabecular bone and cortical bone were applied. The mechanical properties of the materials were assumed to be homogenous, orthotropic, and linearly elastic. And the material properties of the implant-abutment system was set as Zirconia, Titanium-zirconium alloy, Commercially pure Titanium, Ti-Nb-Zr alloy, Gold alloy, Ti-24Nb-4Zr-8Sn alloy respectively, which can be found in the references[4,8] quoted.

Finite Element Models and Interface Condition

The 3D solid model of implant-jaw bone mass assembled by SolidWorks2017 (SolidWorks, SolidWorks Corp. Concord. MA. USA) is imported into ANSYS workbench software (ANSYS Inc. Southpointe. Canonsbury. PA. USA). In order to ensure the convergence of the results, reference[9] sets the maximum mesh size of 0.35 mm, and uses the self-adaptive meshing function to divide the mesh. The three-dimensional finite element model of an entity mandibular segment model with implant-abutment system is established (Fig. 2). The base of an entity mandibular segment was considered to be fixed as close to the real case as possible. The implant and screw joint was assumed to be frictional connection with a friction coefficient of 0.5 according to reference[9], while a friction coefficient of 0.4 is set between implant and peri-implant bone tissue. And the other contact was assumed to be bonded. In this study, a static loading of 130N was applied to the 15°to the long axis of the central of dental crown. The screw was subjected to preload of 100N for Simulating initial stage of planting as far as possible. Fig. 2 reveals the finite element analysis (FEA) conditions.

Statistical Analysis

Differences in stress distributions were tested over various implant-abutment system material combination. And the maximum equivalent stress is used as the index to measure the stress distributions for analyzing the cortical bone, trabecular bone, as well as implants, abutments,
screws with titanium alloy, titanium and gold materials. Maximum principal stress is used to measure the stress distribution for the brittleness zirconia implant, zirconia abutment and zirconia screw according to the first strength theory. Differences in maximum stress in cortical bone over various implant-abutment system material combination using multi-factor variance analysis employing materials of implant, materials of abutment and materials of screws as discriminating variable. And further statistical method of multiple comparison were applied among groups with maximum stress peak in bone tissue. Statistical analysis was done with MATLAB software (MathWorks Corp, Natick. MA. USA) to test the effect of different variables on the maximum stress peak.

Results

The detailed stress distribution contours of the implant, abutment, screw, cortical bone and trabecular bone in these models are shown in Fig. 3. In the FEA, all models of this study was solely based on one geometry to ensure identical test condition.

The first thing to notice is that the stress distribution was nearly identical for all of the models with different material combinations in group A, B, C, D, E, F. The maximum stresses observed in the contact position between the implant and the abutment and the implant stress was concentrated in the upper threads. The maximum stress of the screw appeared in the first threads, while and the stress of the bottom threads of the screw was smaller. It is clear from the Fig. 3 that the stresses distribution in cortical bone, most notably the region adjacent to the implant neck, were greater than those in trabecular bone. In addition, a relatively small amount of stress appeared in the trabecular bone, particularly appeared in upper areas around implant neck.

Multi-factor variance analysis of these interactions is summarized in Table 1. The results showed that implant materials and abutment materials had significant effects on the maximum stress peak of cortical bone (P < 0.05). The parameters of screw materials had no significant effects on the maximum stress peak of cortical bone (P > 0.05). The interaction between implant materials and abutment materials had significant effects on the peak stress of cortical bone (P < 0.05). The interaction between screw and implant materials had no significant effects on the maximum stress peak of cortical bone (P > 0.05). And there was no statistically significant difference between the abutment materials and the selection of screw materials (P > 0.05).
Table 1. Summary of the analysis of variance (ANOVA) showed the statistical results of maximum stress peak with respect to cortical bone.

<table>
<thead>
<tr>
<th>Source</th>
<th>Sum Sq</th>
<th>DF</th>
<th>Mean Sq</th>
<th>F</th>
<th>Prob&gt;F</th>
</tr>
</thead>
<tbody>
<tr>
<td>Implant Material</td>
<td>743.99</td>
<td>5</td>
<td>148.797</td>
<td>223.34</td>
<td>0.0000</td>
</tr>
<tr>
<td>Abutment Material</td>
<td>648.69</td>
<td>5</td>
<td>129.738</td>
<td>194.73</td>
<td>0.0000</td>
</tr>
<tr>
<td>Screw Material</td>
<td>6.06</td>
<td>5</td>
<td>1.211</td>
<td>1.82</td>
<td>0.114</td>
</tr>
<tr>
<td>Implant Material*Abutment Material</td>
<td>52.16</td>
<td>25</td>
<td>2.086</td>
<td>3.13</td>
<td>0.0000</td>
</tr>
<tr>
<td>Abutment Material*Screw Material</td>
<td>25.96</td>
<td>25</td>
<td>1.038</td>
<td>1.56</td>
<td>0.0588</td>
</tr>
<tr>
<td>Implant Material*Screw Material</td>
<td>15</td>
<td>25</td>
<td>0.6</td>
<td>0.9</td>
<td>0.6037</td>
</tr>
<tr>
<td>Error</td>
<td>83.28</td>
<td>125</td>
<td>0.666</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Total</td>
<td>1575.13</td>
<td>215</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

*Multi-factor variance analysis (significant difference for P<0.05). Sum Sq, Sum of squares; DF, degrees of freedom; Mean Sq, mean square.

The minimum stress mean value of cortical bone and trabecular bone were measured in group F model, which used Ti2448 as implant material. The main results of the multiple comparison in maximum stress peak of cortical bone and trabecular bone were summarized in Fig. 4. Which showed the difference of the maximum stress peak value of cortical bone and trabecular bone between different implant material groups and the circle shows the mean maximum stress peak value of cortical bone and trabecular bone.

These trend of maximum stress peak value of abutment are close to each other. And young’s modulus showed negative correlation with maximum stress peak value of abutment (see Fig. 5). In this context, the grater young’s modulus of the abutment with a smaller the maximum stress peak value.

Discussion

The literature has shown that FEM has been widely used in oral biomechanical analysis because of
its non-invasive, high efficiency and repeatability. Proper stress distribution and stress value of dental implant-abutment system and peri-implant bone tissue can prevent most implant failures.

As showed in Fig. 4, the use of different material in implant-abutment system that influence the maximum stress peak value of bone tissue around implant. The stress of dental materials selected in this experiment did not reach the yield limit of materials, while the stress of bone tissue around implants remained in a relatively stable range. The mean maximum stress peak value of cortical bone in 6 groups are ranged from 45.65 to 51.62 Mpa, and the mean maximum stress peak value of trabecular bone are from 4.79 to 5.16 Mpa. Despite this, the results in stress illustrate that the peak stress of cortical bone is 10 times that of trabecular bone. And it is worth emphasizing that if the stress decrease in cortical bone and increase in trabecular bone due to the occlusal force transference, the implant-abutment system will have better stress dispersion effect[10].

The young’s modulus of the material has long been a topic of research in the implant-abutment system, because the phenomenon called stress shielding. As the young’s modulus of implant and peri-implant bone tissue are different, the material of implant with large young’s modulus will bear more stress than surrounding bone tissue within the transmission process of occlusal force. A possible reason for the gradual atrophy of the bone is that this stress shielding reduce stimulation in cortical bone and trabecular bone[6]. During this FEA, Table 1 and Fig. 4 showed the material of implant influence the stress of surrounding bone.

Various young’s modulus did cause the maximum stress peak value of abutment changed(Fig. 5), negative correlation was found between the young’s modulus of abutment and the maximum stress peak value of abutment expect implant material was Ti2448. The possible rationale is that there have interactions between implant and abutment materials. Excessive difference of young’s modulus between implant and abutment material, otherwise known deformation, also contribute to difference stress. As showed in Table 1, the maximum stress peak value of cortical bone may be associated with implant material and abutment material. Consequently, optimal implant material design need to consider the compatibility with abutment and screw materials. Suitable material combinations of implant-abutment system will obtain better stress dispersion effect for minimizing the maximum stress peak value of cortical.

Conclusions
From the obtained results it was possible to conclude that:
(1) Different implant material play a role for the mean of maximum stress peak value in peri-implant bone tissue.
(2) The maximum stress peak value of cortical bone may be associated with abutment material and implant material. And optimal implant-abutment system material design need to consider the compatibility with abutment and screw materials.

Acknowledgement
This study was supported by the Innovation projects of Department of Education of Guangdong Province, China (No. 2017KTSCX183) and the introduction project of Jiangmen innovative research team (2018).

References


