Stress distribution of an internal connection implant prostheses set: A 3D finite element analysis


SUMMARY

Objective. This study evaluated the stress distribution by loads, under an internal connection implant system with two sorts of abutment screws and prosthetic crown models at ten observation points.

Materials and method: The analysis were made in two models with internal butt joint, and with gold and titanium screw, respectively. The load was 382N with 90° to the occlusal surface and 15° to the implant axis at 4 and 6 mm from the implant center.

Results: In both models, a large amount of stress was located around the implant neck and little stress was concentrated along the abutment screw.

Conclusion: The simulations made suggest that the internal connection protects the abutment screw from the accumulated stresses; however, it exposes the implant walls to these stresses.

Key words: finite element analysis, abutment screw, biomechanic.

INTRODUCTION

Initially, titanium implants were used to rehabilitate completely edentulous patients, referred to as oral invalids, with the goal of reestablishing masticatory function. With the development of techniques and materials used in implant dentistry, other indications were added, and now implants are used to rehabilitate cases of partial and single edentulism [1,2], combining function and esthetics.

In time and with the increasing use osseointegrated implants, a number of biomechanics problems began to appear [3]. In an endeavor to minimize such problems, alternative connection systems, based on the internal opposition of implants and abutments walls were developed, either a cone-shaped connection with an angle ranging from 8 to 11° [4,5], or one with an internal hexagon-like format [6,7]. Hypothetically it could reduce the stress on the abutment screw, because the occlusal stress transferred to the abutment could be divided between the internal joint walls and the abutment screw [7].

However, these changes diminished these biomechanical problems, but did not eliminate them, therefore, further longitudinal studies are required to make the use of these new technologies safer. Many computer and engineering scientists have used virtual models and environments, called finite element analyses, to run simulations and progressively test the resistance and stress distribution of tools and parts of machines used in present day life. According to Geng et al. (2001) [8], mechanical problems are solved by splitting the main problem into small and simple elements, arranged in a net shape, called a mesh, in which the analyzed variables are interpolated in the form mathematical functions.

The aim of this preliminary study was to make a qualitative comparison and evaluation, by means of 3D-finite element analysis, the stress distribution produced by simulated load under an internal connection implant system with two sorts of abutment screws and prosthetic crown models at ten observation points.
In this study, the following components and materials were used: 2 Osseotite Certain implants measuring 11.5×4.0 mm in size (3i, Implant Innovations, Palm Beach, Florida, USA), 2 Gengihue abutments of 2 mm (3i, Implant Innovations, Palm Beach, Florida, USA), 2 gold screws, Gold Tite® hexagonal head (3i, Implant Innovations, Palm Beach, Florida, USA). By exposing the test sets to a Profile Projector (Nikon model v.16, serial number 36914, RBC Certificate 7463.1) it was possible to develop two 3D virtual models. Model 1 and 2 with pre-load of 20 N/cm², applied on the contact surface of the screw head with the respective abutment, consisting of: model 1; (Osseotite Certain implant measuring 11.5×4.0 mm in size, Gengihue abutment of 2 mm, hexagonal gold abutment screw, Ag-Pd metal coping); model 2; (Osseotite Certain implant measuring 11.5×4.0 mm in size, Gengihue abutment of 2 mm, hexagonal titanium abutment screw, Ag-Pd metal coping) by means of the CAD computational tool (PRO-ENGINEER, PTC, Needham, MA, USA). The coronal end of the models was extracted from a real dental element, through Computer Tomography of a human tooth (46 dental element) due to the geometric complexity of the occlusal surface and imported to the PRO-E System. Models 1 and 2 were divided into reduced solid elements, originating 8277 tetrahedral elements each, giving rise to a virtual net called mesh (Figure 1). The intrinsic material characteristics used to construct the models are listed in Table 1. Next, virtual simulations of load applications on the models were done by the PRO-MECHANIC System, also developed by PTC, analyzing and comparing the stress distribution induced by the loads applied on the model structure by Von Misses Stress (EQV-MPa), the stresses were analyzed at ten different observation points (Table 2). A static load of 382N was applied, simulating loads in the molar region [9] and was applied parallel to and at a 15° inclination to the implant axis [10] in two different regions of the superior surfaces of metal ceramic crown models, at 4 mm and 6 mm from the center of the implant model [11]. To enhance evaluation of the results, some restrictions were placed on the implant model movements, simulating a vertical bone loss level of 3 mm from the implant platform.

**MATERIALS AND METHOD**

In this study, the following components and materials were used: 02 Osseotite Certain implants measuring 11.5×4.0 mm in size (3i, Implant Innovations, Palm Beach, Florida, USA), 02 Gengihue abutments of 2 mm (3i, Implant Innovations, Palm Beach, Florida, USA), 02 gold screws, Gold Tite® hexagonal head (3i, Implant Innovations, Palm Beach, Florida, USA). By exposing the test sets to a Profile Projector (Nikon model v.16, serial number 36914, RBC Certificate 7463.1) it was possible to develop two 3D virtual models.

Model 1 and 2 with pre-load of 20 N/cm², applied on the contact surface of the screw head with the respective abutment, consisting of: model 1; (Osseotite Certain implant measuring 11.5×4.0 mm in size, Gengihue abutment of 2 mm, hexagonal gold abutment screw, Ag-Pd metal coping); model 2; (Osseotite Certain implant measuring 11.5×4.0 mm in size, Gengihue abutment of 2 mm, hexagonal titanium abutment screw, Ag-Pd metal coping) by means of the CAD computational tool (PRO-ENGINEER, PTC, Needham, MA, USA). The coronal end of the models was extracted from a real dental element, through Computer Tomography of a human tooth (46 dental element) due to the geometric complexity of the occlusal surface and imported to the PRO-E System. Models 1 and 2 were divided into reduced solid elements, originating 8277 tetrahedral elements each, giving rise to a virtual net called mesh (Figure 1). The intrinsic material characteristics used to construct the models are listed in Table 1. Next, virtual simulations of load applications on the models were done by the PRO-MECHANIC System, also developed by PTC, analyzing and comparing the stress distribution induced by the loads applied on the model structure by Von Misses Stress (EQV-MPa), the stresses were analyzed at ten different observation points (Table 2). A static load of 382N was applied, simulating loads in the molar region [9] and was applied parallel to and at a 15° inclination to the implant axis [10] in two different regions of the superior surfaces of metal ceramic crown models, at 4 mm and 6 mm from the center of the implant model [11]. To enhance evaluation of the results, some restrictions were placed on the implant model movements, simulating a vertical bone loss level of 3 mm from the implant platform.

**RESULTS**

Comparing the two types of screws, gold and titanium, in models 1 and 2, a clear balance was shown in the stress values of the ten observations points (Figure 2). This balance is stronger when direct comparison was made between the abutment screw points (M3’c, M4c, M6c, M7c, M8c, M9c). In these models (1 and 2) there was no linear behavior related to the load application point (Tables 3 and 4). In both models, a large amount of stress was located around the implant neck (Figures 3 and 4) and little stress was concentrated along the abutment screw (Figures 5 and 6).

![Fig. 1. Schematic image of model mesh configurations](image-url)

**Table 1. Characteristics of the materials used for simulating the models**

<table>
<thead>
<tr>
<th>Models</th>
<th>Modulus of elasticity (mpa)</th>
<th>Poisson’s ratio</th>
<th>References</th>
</tr>
</thead>
<tbody>
<tr>
<td>Ti (ASTM-F67)</td>
<td>100 x 10³</td>
<td>0.34</td>
<td>Akour et al (2005)[24]</td>
</tr>
<tr>
<td>Ti6Al4V (ASM-TM-F-136)</td>
<td>110 x 10³</td>
<td>0.34</td>
<td>Akour et al (2005)[24]</td>
</tr>
<tr>
<td>Gold alloy-type 3</td>
<td>100 x 10³</td>
<td>0.30</td>
<td>Geng et al (2001)[8]</td>
</tr>
<tr>
<td>Feldspatic ceramic</td>
<td>68.9 x 10³</td>
<td>0.28</td>
<td>Geng et al (2001)[8]</td>
</tr>
<tr>
<td>Ag-Pd Alloy</td>
<td>95 x 10³</td>
<td>0.33</td>
<td>Geng et al (2001)[8]</td>
</tr>
</tbody>
</table>

**Table 2. Points of observation – models 1 and 2**

<table>
<thead>
<tr>
<th>M0c</th>
<th>Abutment/Implant intersection – load application side</th>
</tr>
</thead>
<tbody>
<tr>
<td>M1c</td>
<td>Abutment/Implant intersection – side opposite to load</td>
</tr>
<tr>
<td>M2c</td>
<td>Internal point of hexagon – load application side</td>
</tr>
<tr>
<td>M3c</td>
<td>Internal point of hexagon – side opposite to load</td>
</tr>
<tr>
<td>M3’c</td>
<td>Half of internal diameter of 1st thread of screw on side opposite to load</td>
</tr>
<tr>
<td>M4c</td>
<td>Half of internal diameter of 1st thread of screw on side opposite to load</td>
</tr>
<tr>
<td>M6c</td>
<td>2nd thread of screw – load application side</td>
</tr>
<tr>
<td>M7c</td>
<td>2nd thread of screw – side opposite to load</td>
</tr>
<tr>
<td>M8c</td>
<td>Half of internal diameter of 2nd thread of screw on load application side</td>
</tr>
<tr>
<td>M9c</td>
<td>Half of internal diameter of 2nd thread of screw on side opposite to load</td>
</tr>
</tbody>
</table>
DISCUSSION

The biomechanics of the implant-screw-abutment complex is very differentiated. As the concepts of osseointegration began to consolidate, this region of the implant-prosthesis set started arousing interest, because the mechanical problems, such as loosening of prosthetic abutment screws, their fractures, and sometimes implant fractures that had long intrigued clinicians, began to occur more frequently, when compared with problems related to osseointegration. According to Binon et al. (1994), these problems might be caused, mainly due to inadequate torque, prostheses lacking adaptation and passive fit [12], occlusal overload and unsuitable retainer screw design [13]. According to Wiskott et al. (2004) there is a directly proportional relationship between the preload applied on abutment screws and their resistance to fatigue, which may cause severe mechanical problems [14].

In the assessment made by Goodacre et al. (2003) through a systematic literature review, a large number of mechanical complications have been reported recently, and of the articles reviewed by the author, 1% of them showed the presence of fistula at the level of the prosthetic connection, 6% indicated loss of abutment screws, 45% of these losses being in unit crowns; abutment screw fractures occurred in 2% of the studies and implant fractures in 1% [15].

The use of the finite element method to analyze stress concentrations was initially introduced into implant dentistry by Weinstein et al. (1976) [16]. The models used can be bidimensional [11, 16] or three dimensional [17, 18]. The analyses can be done from the bone-implant interface point of view, relating stress concentrations and displacements between titanium and bone [19, 20, 21, 22, 23] in the implant-screw-abutment set relating the geometric form of the prosthetic connections and screw materials with potential risks for failure [24, 25] or suggesting modifications in im-
plant and prosthetic component designs to maximize clinical performance [26, 27].

The majority of the studies related to FEA in the implant connection – prosthesis used loads of magnitude ranging between 35N [25] and 80N [24], which justify the high values found in this study, since the load used was 382N, around 4 times the value of the highest load found in the literature. Although laboratory and clinical studies [3, 5, 28, 14, 29] used cyclic loads related to the causes of loss of abutment screws with dynamic fatigue, the majority of studies revised, which used FEA in their methodology, used static loads to facilitate stress analysis, since more complex virtual tests require materials considered anisotropic, non linear and heterogeneous [8].

The highest and lowest accumulated stress values in models 1 and 2 were: 5.256543E+02 MPa, 5.25544E+02 MPa and 1.548951E+01 MPa, 1.808135E+01 MPa, respectively. According to Jörnén et al. (1992), the only screw capable of maintaining a single implant supported restoration stable under extreme load situations, is the gold screw because of the mechanical characteristics of the alloy [30].

Models 1 and 2 demonstrated very closed values when homologous situations were compared, both at the points situated on the screw and those located outside of it. At some points, the stresses found were practically equal, such as for example, at the point of observation M1c, at the half of the internal hexagon face, for which the stress values were 2.442097E+02 MPa for model 1 and 2.442249E+02 MPa for model 2, suggesting that for implant models studied in this paper, the material the abutment screw is made of does not have great influence on the stress distribution along the implant-abutment complex. This can be explained due to the division of the masticatory forces transmitted between the abutment screw and the walls of the
internal prosthetic connection [7, 28], which may prolong the useful life of the abutment screws, however, it could prejudice the integrity of the implant in the long term, and could precipitate fracture by fatigue.

CONCLUSIONS

When analyzing the stress concentration values of internal connection implants in different abutment screws, it was possible to conclude that:

- The type of material the abutment screw is made of did not have influence on the stress distribution along the prosthetic connection.
- It was not possible to identify the most harmful load situation for the studied points, as the behavior of the stress concentrations did not suggest a pattern.
- The simulations made suggest that the internal connection protects the abutment screw from the accumulated stresses; however, it exposes the implant walls to these stresses.

Analysis by finite elements was shown to be a versatile and promising methodology for analyzing stress concentrations in implant dentistry, but it is worth emphasizing that the FEA (Finite Element Analysis) is an approximate virtual simulation of clinical situations, presenting certain limitations.

REFERENCES


Received: 28 03 2007
Accepted for publishing: 19 05 2009