COMPARATIVE MEASUREMENT OF FRACTURE RESISTANCE OF VARIOUS KIND INTERNAL CONNECTION SYSTEMS

THE EUROPEAN ASSOCIATION FOR OSSEOINTEGRATION
COMPARATIVE MEASUREMENT OF FRACTURE RESISTANCE OF VARIOUS KIND INTERNAL CONNECTION SYSTEMS

Abstract
Background:
Dental implants are a well accepted treatment for partially or totally edentulous subjects. Innovations through research have led to advancements in surgical and restorative techniques, improved surface features and restorative components. Dental implants typically use either internal or external connections with the crown. Although both connections are extensively used clinically, distinctly different stress distributions are produced within the crown. Although both connections are extensively used clinically, distinctly different stress distributions are produced within the crown. Clinicians have reported implant components linked to mechanical failure of crown and implant.

Aim:
The purpose of this study was to compare the stress distribution characteristics of different abutment connection systems under occlusal loading, using 3-dimensional (FEM) Finite Element Method.

Methods:
In this study three different implant brands (Dental Implant KA® and other four implants) were investigated. The strain distribution of different implant-abutment connection systems having same material properties were evaluated under same loading conditions, the advantages and disadvantages of each system were assessed. The investigations were performed using FEM (Finite Element Analysis Software) methods in a software-based system.

Results:
No differences were found between different implant brands.

Conclusions and clinical implications:
In this study different Dental Implant KA implant samples were applied by means of finite element computational method. In the conclusion not a significiation effect was observed.

Methods and Materials
The 3-D FEA is considered an appropriate method for investigation of the stress throughout a 3-D structure, and therefore this method was selected for bone and implants stress evaluation in this study. The software SOLIDWORKS (Yenasoğlu & Ağaşehir, Istanbul) was used for preprocessing, finite element analysis, and ANSYS 14.0 for postprocessing in the study. In this study five different implant brands (Dental Implant KA and other four implants) were investigated. The strain distribution of different implant-abutment connection systems having same material properties were evaluated under same loading conditions, the advantages and disadvantages of each system were assessed. The investigations were performed using FEM (Finite Element Analysis Software) methods in a software-based system.

Forces of 300 N, 200 N, 100 N, 50 N and 30 N were separately applied axially (AX), to the center of the internal connection to tally 5 different FE model and the von Mises stresses (equivalent [EQV] stresses) in the structure were calculated.

<table>
<thead>
<tr>
<th>The Course of Force</th>
<th>Vertical</th>
<th>Horizontal</th>
<th>Oblique</th>
</tr>
</thead>
<tbody>
<tr>
<td>The Amount of Force (Newton)</td>
<td>100</td>
<td>30</td>
<td>50</td>
</tr>
<tr>
<td>Maximum Value of Stress (Von misses N/m²)</td>
<td>1.068.58 6.432.0</td>
<td>1.007.786.1 76.0</td>
<td>1.244.403.4 56.0</td>
</tr>
</tbody>
</table>

Table 1: The amount, direction and stress values of the forces used in the 3D Model

Regardless of abutment types and loading conditions, higher stress concentration was observed at the cortical bone. In cancellous bone, the highest stress was observed at apical portion and the maximum stress occurred at the implant neck. The higher internal stress was observed in the fixtures than in the bone. No differences were found between different implant brands. In this study different Dental Implant KA implant samples were applied by means of finite element computational methods. In the conclusion not a signification effect was observed.
Figure 1: The stress distribution on Implant KA after applying angulated 100 N force

Figure 2: The stress distribution on Implant KA after applying angulated 200 N force

Figure 3: The stress distribution on Implant KA after applying angulated 200 N force

Figure 4: The stress distribution on the other implant after applying 300 N force

Figure 5: The dimension of montaged mesh on the other implant

Figure 6: The fragile point of 300N montaged other implant
Conclusions
Since the highest stress was observed at implant neck, the design of an internal connection, that distributes the forces homogeneously around the implant collar, is essential. 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 stimulation of clinical situations presenting certain limitations.

References