The Effect of Bone Type on Peri-implant Bone Stress in Regular and Narrow Diameter Implants on Oblique Loading: A Three-dimensional Finite Element Analysis

ABSTRACT

Purpose: A three-dimensional (3D) finite element analysis (FEA) was performed to evaluate the influence of bone quality, on the stress/strain in bone surrounding the implant.

Materials and methods: Three-dimensional finite element models created to replicate completely osseointegrated endosseous titanium implants, were used for the purpose of stress analysis. Two study groups consisting of a regular platform (RP) group and a narrow platform (NP) group were used with four different bone densities and loaded using the ANSYS workbench software to calculate the Von Mises and principal (maximum tensile and minimum compressive) stress.

Results: Maximum equivalent stress/strain in bone increased with a decrease in cancellous bone density. Under oblique load, especially in the low-density bone models, maximum equivalent strain was lower with the RP implant model than with the NP implant model.

Discussion: This study confirms the importance of bone quality and its presurgical diagnosis for implant long-term prognosis. Implant and abutment diameter can also influence bone strain, especially in low-density bone.

Conclusion: The results of this study suggest that bone of higher rather than lower density might ensure a better biomechanical environment for implants. Moreover, regular diameter implants could be a better choice in a jaw with cancellous bone of low density.

Keywords: Bone dentistry, Dental implants, Finite element analysis, Implant diameter, Implant loading.

How to cite this article: Mary AH, Philip JM, Jain AR, Venkatakrishnan CJ. The Effect of Bone Type on Peri-implant Bone Stress in Regular and Narrow Diameter Implants on Oblique Loading: A Three-dimensional Finite Element Analysis. J Dent Sci Oral Rehab 2016;7(1):1-4.

Source of support: Nil

Conflict of interest: None

INTRODUCTION

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 an important factor, with more failures found in bone of lower density. Bone quality influences the long-term success of implant treatment, with poor bone quality leading to lower success rates. The classification for bone quality (types I to IV bone) proposed by Lekholm and Zarb has been widely applied by clinicians in evaluating patient bone for implant placement. Jaffin and Berman found that only 3% of Brånemark System implants (Nobel Biocare, Göteborg, Sweden) placed in type I, II, and III bone were lost after 5 years, while in type IV bone, failure rates were 35% over the same period. van Steenberghe et al. also found more failures in maxillae with poor bone quality. Since the bone around implants must react to stresses and strains generated by occlusal loads, bone with poor quality could more easily fail to withstand these loads. Clinically, these factors are difficult to investigate because of limited information and sample variation. To verify the hypothesis that bone stress and strain are influenced by bone quality, a three-dimensional finite element analysis (3D FEA) was performed.

MATERIALS AND METHODS

Three-dimensional finite element models created to replicate completely osseointegrated endosseous titanium implants were used for the purpose of stress analysis. The models were constructed using measurements and geometries similar to previous studies, with isotropic material properties (Table 1). An isotropic material is defined as having identical physical properties in all directions; therefore, only two independent material constants exist. A computer aided design (CAD) package called Pro Engineer Wildfire (Parametric Technology corporation, Waltham, MA, USA) was used to generate the models in a.prt file format. Using the Pro/E feature and parameteric-based design capability, the implant critical dimensions, such as the height, diameter, etc. were defined to create a virtual assembly in a mesh form.
Three 3D finite element models were created to replicate an implant (13 mm in length with 0.375 mm V thread depth and 0.6 mm pitch) and abutment (6.5 mm in length) with peri-implant bone tissue in which three different implant-abutment configurations were represented. Two implant abutment models consisting of a regular platform (RP) model where a regular 4.3 mm diameter abutment was connected to regular 4.3 mm diameter implant and a narrow platform (NP) model where a 3.5 mm diameter abutment was connected to a 3.5 mm diameter implant was used. A three-dimensionally generated finite element model of a Ni-Cr restoration of 8 mm height, 8 mm in maximum diameter and with an occlusal thickness of 1.5 mm was designed over the abutment.

Complete osseointegration at the implant-bone interface was simulated by combining the nodes of the implant and bone models. Similar integration of the abutment and implant body was adopted, to be a single unit. The same type of contact was also provided at the prosthesis-abutment interface. This eliminated any potential influence from the micromovement between components.

Each of the two implant abutment models were tested in D1, D2, D3 and D4-bone density environments (Table 1), assumed to be homogeneous, isotropic and linearly elastic. Three dimensionally generated FE models of the bone tissue models in which the two implant-abutment configurations, RP and NP with the prosthesis are embedded was designed to be blocks of 16 mm in height, 11 mm in width and 11 mm in breadth. Oblique load of 90N was applied on the flat surface of the restoration on the abutment of the 3D FEA models (Figs 1 and 2). The oblique loading angle of 35.6° imitated the chewing pattern recorded with a jaw tracking device by Ishigaki et al.

The finite element mesh was generated with the following nodes and elements. The final models had a total of 139,334 nodes and 74,324 elements for the NP model, 113,596 nodes and 65,897 elements for the PS model, and 120,703 nodes and 60,7536 elements for the RP model.

The implant geometries were digitally imported into ANSYS workbench software (Swanson Analysis System, Houston, PA) after converting into .iges file format and used to calculate the Von Mises and principal (maximum tensile and minimum compressive) stress ranges for the bone on implant loading.

The Von Mises stresses were obtained, when each of the implant-abutment models RP and NP along with the restoration, were embedded in bone and subjected to oblique loading.

**RESULTS**

The data obtained from the 3D generated models created using finite element software, makes it possible to compare the stress distribution in the various bone densities surrounding the two implant abutment
The Effect of Bone Type on Peri-implant Bone Stress in Regular and Narrow Diameter Implants on Oblique Loading

Table 2: Effect of platform configuration on Von Mises, maximum, and minimum principal stress concentrations (MPa) in the models under oblique loading

<table>
<thead>
<tr>
<th>Bone and type of stress</th>
<th>NP</th>
<th>RP</th>
<th>Decrease (%)*</th>
</tr>
</thead>
<tbody>
<tr>
<td>D1#</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Von Mises</td>
<td>105</td>
<td>104</td>
<td>0.95</td>
</tr>
<tr>
<td>Max. principal</td>
<td>138</td>
<td>132</td>
<td>4.34</td>
</tr>
<tr>
<td>Min. principal</td>
<td>−125</td>
<td>−121</td>
<td>3.2</td>
</tr>
<tr>
<td>D2#</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Von Mises</td>
<td>118</td>
<td>114</td>
<td>3.38</td>
</tr>
<tr>
<td>Max. principal</td>
<td>142</td>
<td>138</td>
<td>2.81</td>
</tr>
<tr>
<td>Min. principal</td>
<td>−182</td>
<td>−125</td>
<td>31.31</td>
</tr>
<tr>
<td>D3#</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Von Mises</td>
<td>144</td>
<td>142</td>
<td>1.38</td>
</tr>
<tr>
<td>Max. principal</td>
<td>152</td>
<td>148</td>
<td>2.63</td>
</tr>
<tr>
<td>Min. principal</td>
<td>−156</td>
<td>−152</td>
<td>2.56</td>
</tr>
<tr>
<td>D4#</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Von Mises</td>
<td>205</td>
<td>165</td>
<td>19.51</td>
</tr>
<tr>
<td>Max. principal</td>
<td>189.5</td>
<td>178.5</td>
<td>5.80</td>
</tr>
<tr>
<td>Min. principal</td>
<td>−295</td>
<td>−262</td>
<td>11.18</td>
</tr>
</tbody>
</table>

*Decrease in stress calculated as: (NP−RP)/NP × 100%; #D1, D2, D3, and D4-bone density

models during oblique loading. The positive values of the maximum principal stress and the negative values of the minimum principal stress were taken to indicate maximum tensile stress and maximum compressive stress, respectively. To enable comprehension of the effect of bone density on the peri-implant bone stress, the percentage differences in stress values among the groups are shown in Table 2.

When the peri-implant bone tissue was analyzed, cortical bone (D1) exhibited lower stress levels than the trabecular bone (D2, D3, D4) in all models and on oblique loading (Table 2). It was clearly observed that platform switching reduced Von Mises stress values for oblique loads. Compressive stress was higher than tensile stress in all models.

DISCUSSION

The successful use of dental implants has been well-documented, but implant failures are still unavoidable. Implant failures observed after prosthesis delivery are mainly related to biomechanical complications. The mechanisms responsible for biomechanical 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. Unfortunately, these biomechanical aspects are difficult to investigate using solely clinical or experimental approaches with limited information and sample variations.

In all instances, stress/strain values in bone increased with a decrease in cancellous bone density. Low-density bone has low stiffness, generating a significant implant displacement (sinking and tilting under vertical and oblique loads, respectively). This greater displacement led to higher deformation of the bone, and thus to higher stresses and strains in the cortical and cancellous bone, respectively. This result could perhaps be an explanation of the findings in other clinical reports, in which higher failure rates were observed for type IV bone than for types I to III bone.

Bone quality affected strain for both implant abutment models with strain increasing as bone quality decreased. When placing implants in sites of lower bone density, the operator has been encouraged to place longer or wider-diameter self-tapping implants using a conventional drilling technique without countersinking or to use an osteotome technique without drilling. With the use of such osteocompressive procedures, bone density and primary stability of implants may be improved. Extending the healing period prior to prosthesis fabrication may also increase bone density and yield more favorable force transmission.

CONCLUSION

Within the limitations of this study, the following conclusions can be drawn:

• Maximum von Mises, compressive, and tensile stresses in the peri-implant bone were lower in the D1 bone models than in the D4 bone models for both RP and NP implant abutment combinations
• Higher density bone and wider diameter implant abutment combination yielded a positive result with regard to lowering of peri-implant bone stress levels, in healthy as well as compromised bone qualities.

The reduction of the stress concentration at the implant-bone interface area is a favorable development to ensure the continuity of osseointegration.

REFERENCES

Integrated Prostheses: Osseointegration in Clinical Dentistry.

