A Three-Dimensional Finite Element Study to Characterize the Influence of Load Direction on Stress Distribution in Bone Around Dental Implant

Darush Zia Shamami¹,², Alirea Karimi¹,², Borhan Beigzadeh²,*, Shahram Derakhshan², and Mahdi Navidbakhsh¹,²

¹Tissue Engineering and Biological Systems Research Laboratory, School of Mechanical Engineering, Iran University of Science and Technology, Tehran 16846, Iran
²School of Mechanical Engineering, Iran University of Science and Technology, Tehran 16846, Iran

This study aimed to establish a three-dimensional (3D) finite element model for variation of loading direction to identify its effects on the stress distribution generated around an implant and surrounding jawbone under biomechanical consideration. Twenty one 3D finite element cases containing the endosseous titanium implant in type II bone under three different loads i.e., 50, 100, and 200 N, with various directions from 0 to 30 degrees to the vertical axis with the increasing rate of 5 degrees were applied to analyze the stress distribution pattern in the implant and surrounding bone. For data analysis, the von Mises criterion was used. The results revealed that the oblique loading would significantly induce higher interfacial stresses compared to the vertical loading. The highest on von Mises stress was observed in the implant for all the cases. An increasement of the angle of force both increased the maximum stress and worsened stress distribution patterns in the implant and supporting bone. The results suggest that under oblique load 30° the maximum von Mises stresses in the implant, cortical, and cancellous bones increase by 236%, 322%, and 22%, respectively. The findings of this study may have implications not only for understanding the stress distribution in implant and bone under various loading angles but also for determining an optimum implant for specific application in dentistry.

Keywords: Dental Implant, Finite Element Analysis, Stress Analysis, Jawbone, Load Direction.

1. INTRODUCTION

Endosseous dental implants play pivotal role in prosthodontics and restorative dentistry since the early 1970s.¹ Owing to its reliable functional and aesthetic results, dental rehabilitation with implants has been widely accepted by doctors and patients in recent decades.² High implant success rates of the order of 78–100% have been reported, with more than 25 years of observation time.³⁻⁵ Despite these high success rates, implant failure, marginal bone loss, and patient discomfort still sometimes occur.⁶⁻⁸ Generally speaking, the success of dental implants is correlated to the biologic tissues (soft tissue and bone) response and mechanical components strength (implant components and superstructures) as well as surgical technique.⁹,¹⁰ One of the reasons for implant failure is Oclusal overload or fatigue-induced microdamage.¹¹ Implants may also fail due to the loss of bone surrounding the implant.¹² Bone loss usually begins at the crestal area of the cortical bone and can progress toward the apical region, jeopardizing the longevity of the implant and prosthesis.¹³ Stress and strain fields around osseointegrated dental implants are affected by a number of biomechanical factors, including the type of loading, material properties of the implant and the prosthesis, implant geometry, surface structure, quality and quantity of the surrounding bone as well as the nature of the bone-implant interface.¹⁴,¹⁵ Biomechanical factors are important for longevity of osseointegrated implants. Controlling these factors prevents mechanical complications, which include fracture of screws, components, or materials veneering the framework.¹⁶ The application of engineering knowledge in dentistry has contributed to an understanding of biomechanical aspects related to implantology.¹⁷ Finite element analysis (FEA) was initially developed in the early 1960s to solve structural problems in the aerospace industry but has since been extended to implantology.¹⁸ The bone stress distribution pattern is highly relevant to the bone-implant relationship and consequently to its
A 3D Finite Element Study to Characterize the Influence of Load Direction on Stress Distribution in Bone

Shamami et al.

There are some studies which investigated the influence of occlusal overload\textsuperscript{13,20} and oblique forces\textsuperscript{16,21–23} on the bone around dental implants. It has been indicated that overloading and load angles should be considered as the very important factors in loss of osseointegration of dental implants. However, a few studies focused on the load direction and its effects on the stress distribution pattern around dental implants. The purpose of this study is to evaluate the effects of external loading under various directions on the stress distribution in the dental implant and surrounding jawbone using 3D finite element analysis.

2. MATERIALS AND METHODS

The 3D finite element models were built with CATIA V5R19 software and ABAQUS 6.10-1 software (SIMULIA Corporation, Providence, RI, United States) to analyze the stress distributions. The model contained abutment, implant, cortical, and cancellous bones, which three values for load assigned to it, including 50, 100, and 200 N. The model was subjected to loading at various directions from 0 to 30 degrees to the vertical axis of implant with increasing rate of 5 degrees were applied to the system to analyze the stress distribution in the implant and supporting bone. The implant with standard diameter was presented in form of screw and abutment of conical form and adjusted to the implant. The bone was modeled as full structure that is composed of a spongy center surrounded by 2 mm of cortical bone.

2.1. Implant System

A Branemark implant (length and diameter of 13 and 3.9 mm, respectively) and triangular thread type (pitch = 1.25 mm) was connected to conic standard abutment (length = 5 mm, large diameter = 3.9 mm, and small diameter = 3 mm).

2.2. Material Properties

The materials used in the current study were assumed to be homogenous, isotropic, and linearly elastic. The physical properties of different components modeled are listed in Table I. The implant and abutment were all designed to be titanium alloy, with an elastic modulus of 110 GPa. Two types of bone density were modeled by varying the elastic moduli of compact bone and cancellous bone with elastic moduli of 13.7 and 1.37 GPa, respectively.

<table>
<thead>
<tr>
<th>Component</th>
<th>Elastic modulus (GPa)</th>
<th>Poisson’s ratio</th>
<th>Reference</th>
</tr>
</thead>
<tbody>
<tr>
<td>Cortical bone</td>
<td>13.7</td>
<td>0.3</td>
<td>[59]</td>
</tr>
<tr>
<td>Dense cancellous bone</td>
<td>1.37</td>
<td>0.3</td>
<td>[59]</td>
</tr>
<tr>
<td>Titanium</td>
<td>110</td>
<td>0.33</td>
<td>[60]</td>
</tr>
</tbody>
</table>

2.3. Model Design and Interface Conditions

Three values of load, i.e., 50, 100, and 200 N, were applied to each set of the model, such as the abutment and implant, with various directions of load from 0 to 30 degrees to the vertical axis of implant with the increasing rate of 5 degrees. Abutment height of the models was modified to 5 mm from the crown margin to the top of the abutment. Implant was connected to conic standard abutment. The rough surface of the testing fixture with 13 mm length was totally embedded in the bone, and interface conditions between the bone and implant were assumed to be fully osseointegrated. The crown margin to the crestal bone level was designed with the same distance of 2 mm (Fig. 1).

2.4. Model Building and Geometry

The bone model was simplified to a cuboid form (14 mm × 14 mm × 25 mm) and was classified as type II bone; type II bone consisted of a layer of cortical bone with uniform thickness of 2 mm, which was surrounded by a core of dense cancellous bone. The 3D model was built by CATIA computer aided design program, which is capable of input geometric features such as length, angle, diameter, and profile to make drawings of the sample parts and assemblies (Fig. 2).

2.5. Elements and Nodes

Finite element model was executed with one implant system, which contained one type of fixture in combination with one type of bone quality; the numbers of elements and nodes were well refined in the model.
2.6. Finite Element Analyses

The finite element analyses were performed to calculate the von Mises stress distribution. Computer-aided engineering software was used to input a 3D model of the sample and defined the mesh control of the model. After meshing the 3D model, conditions such as loads, constraints, and materials were assigned. Three values of load (50, 100, and 200 N) under various directions were applied in order to analyze the stress distribution in the implant and supporting bone. The loads were applied separately on the center point of the abutment (Fig. 3).

The loads were applied under different directions from 0 to 30 degrees to the vertical axis of implant with the increasing rate of 5 degrees in XY plane.

![Fig. 2](image)

**Fig. 2.** Plane diagram illustrating the model design and geometry of the implant with the rough surface totally embedded in bone. Key: yellow = implant and abutment; green = cortical bone; blue = cancellous bone.

The refined mesh of the abutment and implant were all set to an element size of 0.5 mm, whereas the crestal cortical and cancellous bones were all set to an element size of 1.3 mm.

The finite element analyses were performed to calculate the von Mises stress distribution. Computer-aided engineering software was used to input a 3D model of the sample and defined the mesh control of the model. After meshing the 3D model, conditions such as loads, constraints, and materials were assigned. Three values of load (50, 100, and 200 N) under various directions were applied in order to analyze the stress distribution in the implant and supporting bone. The loads were applied separately on the center point of the abutment (Fig. 3).

![Fig. 3](image)

**Fig. 3.** The loads were applied under different directions from 0 to 30 degrees to the vertical axis of implant with the increasing rate of 5 degrees in XY plane.

### Table II. Maximum von Mises stress (Mpa) in different components of model under a 50-N force with various loading angles.

<table>
<thead>
<tr>
<th>Loading angle (degree)</th>
<th>0</th>
<th>5</th>
<th>10</th>
<th>15</th>
<th>20</th>
<th>25</th>
<th>30</th>
</tr>
</thead>
<tbody>
<tr>
<td>Implant</td>
<td>14.43</td>
<td>20.44</td>
<td>26.45</td>
<td>32.34</td>
<td>38.02</td>
<td>43.44</td>
<td>48.53</td>
</tr>
<tr>
<td>Cortical bone</td>
<td>2.04</td>
<td>3.31</td>
<td>4.45</td>
<td>5.56</td>
<td>6.63</td>
<td>7.65</td>
<td>8.61</td>
</tr>
<tr>
<td>Cancellous bone</td>
<td>0.457</td>
<td>0.462</td>
<td>0.464</td>
<td>0.463</td>
<td>0.466</td>
<td>0.515</td>
<td>0.559</td>
</tr>
</tbody>
</table>

### Table III. Maximum von Mises stress (Mpa) in different components of model under a 100-N force with various loading angles.

<table>
<thead>
<tr>
<th>Loading angle (degree)</th>
<th>0</th>
<th>5</th>
<th>10</th>
<th>15</th>
<th>20</th>
<th>25</th>
<th>30</th>
</tr>
</thead>
<tbody>
<tr>
<td>Implant</td>
<td>28.85</td>
<td>40.89</td>
<td>52.91</td>
<td>64.69</td>
<td>76.05</td>
<td>86.87</td>
<td>97.08</td>
</tr>
<tr>
<td>Cortical bone</td>
<td>4.08</td>
<td>6.63</td>
<td>8.90</td>
<td>11.12</td>
<td>13.26</td>
<td>15.30</td>
<td>17.22</td>
</tr>
<tr>
<td>Cancellous bone</td>
<td>0.914</td>
<td>0.923</td>
<td>0.927</td>
<td>0.926</td>
<td>0.932</td>
<td>1.029</td>
<td>1.119</td>
</tr>
</tbody>
</table>

### 3. RESULTS

The stress field in each case with different loading angles was investigated carefully. The maximum von Mises stresses obtained in the implant and surrounding jawbone with applied load directions from 0 to 30 degrees are shown in Tables II, III, and IV for 50, 100, 200 N force, respectively. The highest maximum von Mises stress was observed in the implant for all the cases. It can be seen that stress concentrations occurred at the thread tips of the implant in the neck region near the cortical bone. In the surrounding jawbone, the maximum effective stress under all loading directions occurred at the regions adjacent to the first thread of implant. The maximum effective stress induced by the oblique load was much higher than that of the maximum effective stress caused by an equal amount of vertical load. Furthermore, an increase in the angle of force application caused not only an increase in maximum stress values but also worsened the stress distribution patterns in the implant and supporting bone. In the other words, the angle of force application had a significant effect on both the contour patterns and the magnitude of stress fields. Under an axial loading condition, the stress fields showed a symmetric pattern, whereas they exhibited an asymmetric pattern in the case of oblique loading (Figs. 4–6). The results showed that under oblique load (30°) the maximum von Mises stresses in the implant, cortical, and cancellous bones increase by 236%, 322% and 22%, respectively. Moreover, the maximum stresses in the implant and surrounding jawbone as a function of loading angle are shown in Figure 7 for all simulated cases. It is clear that the maximum stress in the cancellous bone remained almost constant with increasing angles of force application, exhibiting insensitivity to loading direction. However, the maximum stress in the implant and cortical bone significantly changed with varied loading angles.
Table IV. Maximum von Mises stress (Mpa) in different components of model under a 200-N force with various loading angles.

<table>
<thead>
<tr>
<th>Loading angle (degree)</th>
<th>0</th>
<th>5</th>
<th>10</th>
<th>15</th>
<th>20</th>
<th>25</th>
<th>30</th>
</tr>
</thead>
<tbody>
<tr>
<td>Implant</td>
<td>57.71</td>
<td>81.76</td>
<td>105.83</td>
<td>129.38</td>
<td>152.09</td>
<td>173.74</td>
<td>194.14</td>
</tr>
<tr>
<td>Cortical bone</td>
<td>8.17</td>
<td>13.25</td>
<td>17.81</td>
<td>22.24</td>
<td>26.52</td>
<td>30.59</td>
<td>34.45</td>
</tr>
<tr>
<td>Cancellous bone</td>
<td>1.829</td>
<td>1.846</td>
<td>1.854</td>
<td>1.853</td>
<td>1.863</td>
<td>2.058</td>
<td>2.238</td>
</tr>
</tbody>
</table>

They increased almost linearly with an increase in the loading angle up to 30 degrees.

4. DISCUSSIONS

Due to the complex geometry of bone and dental implant system the closed form solutions in stress analysis is not feasible. The finite element method has been increasingly used in this field to analyze the stress distribution patterns in the implant-bone interface for different root-form implant designs. Overloading and disadvantageous stress distribution pattern in jawbone and implant can cause resorption of the bone and failure of the implant. Therefore, investigation of stress field in implant/jawbone systems is of vital importance. In the present work, an implant/jawbone system in twenty one cases was numerically studied, and the effects of load direction on stress values and distribution patterns in the implant and supporting bone were investigated. Figures 4–6 display several typical contour patterns of von Mises stress and local stress distributions in regions in which the highest stress occurred in the implant and supporting bone under 100 N force and angles of 0, 15, and 30 degrees, respectively. All the results demonstrated that a higher loading angle will result in an increasing the magnitude and worsens the distribution pattern of stress. As reported in the literature, an increased angle of force application increases the stress concentrations in the neck region and thus increases the maximum stress values in the jawbone. Similar results...
were obtained in the cortical and cancellous bones in the present work. Moreover, it was seen that an increased loading angle significantly worsened the stress distribution patterns. Figure 7 shows maximum von Mises stresses in different components of model as a function of loading angle for three forces. It was demonstrated that the maximum values of stress in the implant and cortical bone increased almost linearly up to loading angle of 30 degrees. However, the values of maximum stress in cancellous bone remained nearly unchanged up to 30 degrees.

Although interesting results have found, there are still some shortcomings in this study which need to be considered. All materials in this study considered to be homogeneous and isotropic. Nonetheless, dental material may present hyperelastic or viscoelastic mechanical behavior rather than simple elastic one. Therefore, there is a need to conduct an experimental study using uniaxial or biaxial tensile test machine to measure their anisotropic visco-hyperelastic behavior.

In general, an oblique loading angle is the most severe loading condition that significantly increases the stress values and worsens the distribution patterns. Therefore, it is highly recommended to be avoided as much as possible.

5. CONCLUSION

Based on the results from numerical analyses, the loading direction had a significant effect on the maximum stress values and stress distribution patterns in the implant/bone system. Increases in the angle of force application not only can increase the maximum stress values but can worsen the stress distribution patterns in the implant and supporting bone. The maximum stress in the implant and cortical bone changed greatly with varied loading angles. They increased almost linearly with an increase in the loading angle up to 30 degrees. Whereas, the maximum stress in the cancellous bone remained almost unchanged with increasing angles of force application, exhibiting insensitivity to loading direction.

Conflicts of Interest

The authors declare that they have no conflicts of interest.

Acknowledgments: The authors acknowledge the Iran University of Science and Technology for funding this project.

References and Notes

A 3D Finite Element Study to Characterize the Influence of Load Direction on Stress Distribution in Bone

Shamani et al.


Shamami et al. A 3D Finite Element Study to Characterize the Influence of Load Direction on Stress Distribution in Bone


Received: xx xxxx xxxx. Accepted: xx xxxx xxxx.