Stress and Strain Patterns of 1-Piece and 2-Piece Implant Systems in Bone: A 3-Dimensional Finite Element Analysis

Zeev Ormianer, DMD,* Arial Ben Amar, DMD,† Mariusz Duda, DDS,‡ Sharon Marku-Cohen, DMD,§ and Israel Lewinstein, DMD, PhD‖

Purpose: To determine whether 1P and 2P implants with equivalent geometries exhibited stresses and strains differently under applied loading conditions.

Materials and Methods: Design software simulated 1P and 2P implants restored with metal copings and embedded in 3 cylindrical bone block models that varied in dimensions, density, and percentage of bone-to-implant contact. Three-dimensional, finite element analysis simulated occlusal loading. Experiments evaluated stresses and strains relative to implant design and (1) peri-implant bone thickness, (2) cortical bone thickness, (3) magnitude and direction of occlusal loading, and (4) % bone-to-implant contact.

Results: Implants with equivalent dimensions exhibited comparable stresses and strains in all experimental conditions. Implant diameter and periimplant bone thickness influenced stress levels. Only small-diameter (3.0 mm) 1P implants in low-density bone exhibited stress levels that might adversely affect marginal bone stability.

Conclusions: Implant diameter and periimplant bone thickness influenced load distribution in bone, but the type of implant-abutment transition had no significant effect. Small-diameter 1P implants should be limited to dense bone to minimize stress concentrations. (Implant Dent 2012;21:1–00)

Key Words: 1-piece, 2-piece, dental implant, finite element analysis, bone

Statement of Problem: The transition from implant to abutment is solid in 1-piece (1P) and broken in 2-piece (2P) implant designs. This difference may affect occlusal load distribution and marginal bone response.

Purpose: To determine whether 1P and 2P implants with equivalent geometries exhibited stresses and strains differently under applied loading conditions.

Materials and Methods: Design software simulated 1P and 2P implants restored with metal copings and embedded in 3 cylindrical bone block models that varied in dimensions, density, and percentage of bone-to-implant contact. Three-dimensional, finite element analysis simulated occlusal loading. Experiments evaluated stresses and strains relative to implant design and (1) peri-implant bone thickness, (2) cortical bone thickness, (3) magnitude and direction of occlusal loading, and (4) % bone-to-implant contact.

Results: Implants with equivalent dimensions exhibited comparable stresses and strains in all experimental conditions. Implant diameter and periimplant bone thickness influenced stress levels. Only small-diameter (3.0 mm) 1P implants in low-density bone exhibited stress levels that might adversely affect marginal bone stability.

Conclusions: Implant diameter and periimplant bone thickness influenced load distribution in bone, but the type of implant-abutment transition had no significant effect. Small-diameter 1P implants should be limited to dense bone to minimize stress concentrations. (Implant Dent 2012;21:1–00)

Key Words: 1-piece, 2-piece, dental implant, finite element analysis, bone

Reprint requests and correspondence to: Zeev Ormianer, DMD, Department of Oral Rehabilitation, the Maurice and Gabriela Goldschleger School of Dental Medicine, University of Tel Aviv, Tel Aviv, Israel. Phone: +972-3-6124242, Fax: +972-3-6124226, E-mail: drzeev@ormianer.com

© 2012 by the American Academy of Implant Dentistry. All rights reserved. DOI: 10.1097/ID.0b013e31823fce22

Copyright © Lippincott Williams & Wilkins. Unauthorized reproduction of this article is prohibited.
ies using 3-dimensional (3D) finite element analysis (FEA) documented major variations in the abilities of different implant designs to resist and distribute vertical and lateral occlusal loads in bone. In FEA studies, microstrain (\(\mu e\)) is widely used as a method of measuring the load applied to bone as percent the tissue deformation. Frost hypothesized that specific microstrain levels (thresholds) in bone tissue trigger biologic mechanisms that cause bone to either atrophy (<200 \(\mu e\)), maintain a balanced steady state (200–2500 \(\mu e\)), hypertrophy (2500–4000 \(\mu e\)), or suffer pathological overload (>4000 \(\mu e\)) that may result in bone resorption. Similar thresholds for pathological overload with secondary bone resorption have been reported by Sugiura et al.22

Although contemporary 1- and 2P implant systems may have a similar gross external geometries, internal variations may result in very different patterns of load distribution. For example, it is currently unknown whether the interfacial break between the implant and abutment of 2P systems may enhance or reduce stress concentrations in the crestal bone region compared with 1P implant systems, which have a solid transition between the components.

This study reports on a 3D FEA study that compared load distribution patterns of simulated 1P and 2P implants with different diameters in various bone models. The null hypothesis of this study was that there would be no significant difference in load distribution between 1P and 2P dental implants under conditions of simulated loading. An alternate hypothesis was that 1P implants would exhibit higher stress concentrations than 2P implants in the crestal bone region, but that their resulting strain values would still fall within normal limits of 200 to 4000 \(\mu e\) for crestal bone maintenance.

### Materials and Methods

Engineering design software (SolidWorks Professional 2006; SolidWorks Corp, Concord, MA) and 3D FEA code (ANSYS Workbench 11.0; ANSYS Inc., Canonsburg, PA) were used in a personal computer (XPS 210 Desktop; Dell, Round Rock, TX) with Microsoft Windows XP Professional (Microsoft Corp, Redmond, WA) software to create all digital study models. Each model consisted of either a simulated 1P (Zimmer 1-piece Implant; Zimmer Dental Inc., Carlsbad, CA) or 2P (Tapered Screw-Vent Implant and Hex-Lock Abutment; Zimmer Dental Inc.) implants embedded in 1 of 3 cylindrical bone blocks that differed in characteristics and uniform periimplant thicknesses (t) (Table 1). Implant components were modeled with an elastic modulus of 110 GPa and a Poisson’s ratio of 0.34 GPa, which are consistent with titanium alloy, and were positioned in accordance with the manufacturer’s instructions for placement. Bone-to-implant contact (BIC) formed an interfacial bond between the modeled implants and bone blocks, but the percentage of BIC varied according to the experiment. A simulated titanium alloy coping was attached to each abutment to ensure the uniform transfer of applied load.

The finite element models consist of 4-node tetrahedron elements (consistent element size of 0.5 mm [0.020 in] for all assemblies); which total number of elements varied depending on the implant assembly size.

For the boundary conditions, the outer surface of the bone block was modeled as a fixed, and all of interfaces between components (ie, abutment, screw) were modeled as bonded except the interface between implant and abutment that was modeled as a sliding contact. Top portion of a 1P implant (outside of the bone) is still called abutment in this study though the implant and abutment are 1 piece.

To simulate occlusal loading, 222 N of occlusal force was applied to the implant model at a 30-degree angle (\(\theta = 30\) degrees) with a 1.5 mm buccolingual offset from the vertical axis of the implant. The selected occlusal force represented a slightly higher-than-median value within the range of 140 to 286.7 N, which are mean maximum clenching force values documented in the incisor regions of healthy patients. Offsetting loading at a 30-degree angle was performed in an attempt to approximate normal occlusal patterns during mastication. Before the study, a preliminary experiment evaluated these prescribed boundary and load conditions in the lateral, premolar, and molar locations. The study consisted of 4 experiments that applied the same prescribed boundary and load conditions to simulated sites in the adult maxillary lateral incisor

---

### Table 1. Bone Model Characteristics for FEA Analysis

<table>
<thead>
<tr>
<th>Bone Model</th>
<th>Bone Character</th>
<th>Vertical Thickness</th>
<th>Elastic Modulus</th>
<th>Vertical Thickness</th>
<th>Elastic Modulus</th>
<th>Vertical Thickness</th>
<th>Elastic Modulus</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Cortical</td>
<td>(mm)</td>
<td>(GPa)</td>
<td>(mm)</td>
<td>(GPa)</td>
<td>(mm)</td>
<td>(GPa)</td>
</tr>
<tr>
<td>A</td>
<td></td>
<td>20</td>
<td>3</td>
<td>15</td>
<td>20</td>
<td></td>
<td>3.5</td>
</tr>
<tr>
<td>B†</td>
<td></td>
<td>20</td>
<td>2</td>
<td>15</td>
<td>20</td>
<td></td>
<td>3.5</td>
</tr>
<tr>
<td>C‡</td>
<td></td>
<td>20</td>
<td>2</td>
<td>15</td>
<td>20</td>
<td></td>
<td>3.5</td>
</tr>
</tbody>
</table>

* Uniform periimplant bone thickness.
† Generally representative of D1 or D2 bone.
‡ Generally representative of D2 or D3 bone.
implants. The study measured the government on approximately 3000

When facial plate thickness was/h11350/h11350

frequency of tooth loss/h26 in the aesthetic area of greatest aesthetic need because

region. This location represented the area of greatest aesthetic need because of its visual prominence and having the second-highest documented frequency of tooth loss/h26 in the aesthetic zone. The 4 experiments evaluated the influences of implant design (1P or 2P) and diameter (3.0, 3.7, 4.1, 4.7, or 6.0 mm) on bone stress levels relative to 1 or more secondary variables.

Experiment 1 investigated the influence of periimplant bone thickness on stress levels in homogenous bone (Fig. 1). This experiment evaluated the findings of a prospective, multicenter clinical study/h27 conducted by the US government on approximately 3000 implants. The study/h27 measured the distance from the inner walls of newly prepared implant osteotomies to the outer surfaces of the residual buccal plates, and then correlated residual facial plate thicknesses to future facial bone loss and implant failure. Below the critical threshold of 2 mm facial bone thickness, facial plate resorption and implant failure rates increased.27 When facial plate thickness was ≥2

mm, however, implant survival rates increased and some evidence of marginal bone gain was noted.27 In this study, bone model A (Table 1) was used and osseointegration was assumed to be complete (BIC = 100%). The isotropic, homogeneous elastic modulus of the bone model was based on documented/h29,30 modula of 15 GPa for cortical bone (E1) and 1.5 GPa for trabecular bone (E2). Calculations of average elastic modulus utilized volume fractions of cortical bone (ν1 = 15%) and trabecular cancellous bone (ν2 = 85%) where ν1, ν2, E1, and E2 are volume fractions and elastic modulus of cortical and cancellous bone, respectively:

E = ν1E1 + ν2E2  (1)

A 2-time decay model was used to describe the FEA analysis data presented in Eq. (2) where C1 and C2 (MPa) are constants dependent on the implant assembly size and platform, t (mm) is the bone thickness surrounding the implant in mm, and τ1 and τ2 (mm) are decay constants:

\[ \sigma(t) = C_1 \exp\left(-t/\tau_1\right) + C_2 \exp\left(-t/\tau_2\right) \]  (2)

Experiment 2 investigated the influence of cortical bone vertical thickness. To simulate 2 different cortical bone thicknesses, the first and second segments of this experiment used bone models B and C (Table 1), respectively (Fig. 1). In both segments, osseointegration was assumed to be complete (BIC = 100%). In experiment 3, the influence of occlusal load magnitude and direction served as the secondary experimental variable. Bone model B (Table 1) was used, and osseointegration was assumed to be complete (BIC = 100%). Vertical compressive loading at a 0-degree angle (θ = 0 degrees) from the long axis of the implant model was also applied as part of the stress distribution analysis. The linear elastic model used in this study was expected to cause a linear increase in the maximum stress in the bone.

Experiment 4 investigated how the percentages of BIC (% BIC) influenced stress levels in bone model B (Table 1). BIC was 100% in the cervical and apical regions of the implant models, but ranged from 8% to 100% in the thread regions to approximately <100% osseointegration that really occurs in vivo, regardless of implant design.31 This phenomenon of osseointegration is attributable, in part, to the fact that bone contains numerous narrow spaces that are not mineralized.32

**RESULTS**

Implant design, load (magnitude and direction), boundary conditions, and the quality, mechanical properties, and cortical thickness of bone all influenced bone stress levels in this study. Maximum bone stress levels for both 1P and 2P models in the preliminary experiment were below the yield stress of bone (180 MPa), regardless of placement region (incisor, premolar, and molar). Nearly all recorded stresses and strains in this study were <55 MPa and 3690 με, respectively, for both 1P and 2P implants, regardless of loading conditions or bone
types. These values were less than the critical thresholds of stresses for bone resorption (60 MPa) and microstrain values for bone resorption (4000 με).

In experiment 1, all maximum bone stress levels were 40 MPa or lower in 2-mm thick periimplant bone (Table 2). As periimplant bone thickness decreased to 1 mm, the standard diameter (3.7 mm) 1P design exhibited significantly higher stress distributions (42 MPa) than the 2P design (37 MPa) in the same diameter (Table 2). This difference diminished and stress distribution levels relatively converged between the 2 standard diameter designs as bone thickness increased to 2 mm or more (Table 2). In contrast, there was no difference in stress distribution between wide diameter (4.7 mm) 1P and 2P designs, regardless of periimplant bone thickness (Table 2).

Bone stress levels increased as periimplant bone thickness decreased for all implant designs and diameters (Table 2). For example, maximum bone stress levels for a 2P implant, 3.7 mm in diameter, ranged from 26 to 37 MPa as periimplant bone decreased from 2.5 to 1.0 mm, respectively (Table 2). In contrast, increasing implant diameter reduced bone stress levels despite a corresponding reduction in periimplant bone thickness. For example, a 1P implant, 3.0 mm in diameter with 2.0 mm of periimplant bone thickness, generated 40 MPa of maximum bone stress levels (Table 2). By increasing the implant diameter to 3.7 or 4.7 mm, periimplant bone thickness decreased to 1.5 and 1.0 mm, respectively, with corresponding reductions in maximum bone stress levels of 34 and 24 MPa (Table 2). Based on 2-time decay models, the rate of change in bone stress levels because of bone thickness was independent of the implant design or diameter (Table 2). The elevated stress levels generated by 1P implants, 3.0 mm in diameter, in periimplant bone <2-mm thick, may adversely affect crestal bone maintenance (Table 2).

In experiment 2, 1P and 2P designs in bone models B (Tables 1 and 3) and C (Tables 1 and 4) exhibited the highest stress contour bands in the crestal bone region, but stresses progressively diminished deeper within the bone. Implants in matching lengths and diameters demonstrated no significant differences in crestal bone stress concentrations, regardless of implant design. Although data showed that 1P and 2P models exhibited a 6 MPa difference in maximum von-Mises stress in 2-mm thick bone for both 3.7- and

---

**Table 2. Influence of Implant Design, Diameter, and Periimplant Bone Thickness on Maximum Bone Stress Levels (MPa) in Bone Model A**

<table>
<thead>
<tr>
<th>Periimplant Bone Thickness (t) (mm)</th>
<th>1-Piece Implants by Diameter†</th>
<th>2-Piece Implants by Diameter†</th>
<th>Difference (1P vs. 2P)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Maximum Bone Stress Levels (MPa)</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>3.0 mm</td>
<td>53</td>
<td>42</td>
<td>17</td>
</tr>
<tr>
<td>3.7 mm</td>
<td>45</td>
<td>34</td>
<td>14</td>
</tr>
<tr>
<td>4.7 mm</td>
<td>40</td>
<td>30</td>
<td>13</td>
</tr>
<tr>
<td>3.7 mm</td>
<td>38</td>
<td>28</td>
<td>12</td>
</tr>
<tr>
<td>3.5 mm</td>
<td>36</td>
<td>26</td>
<td>11</td>
</tr>
<tr>
<td>3.0 mm</td>
<td>35</td>
<td>25</td>
<td>11</td>
</tr>
<tr>
<td>5.0 mm</td>
<td>35</td>
<td>25</td>
<td>11</td>
</tr>
</tbody>
</table>

† FEA analysis using 222 N of load applied at a 30-degree angle and a 1.5 mm offset.

‡ Not recommended for actual clinical applications.

---

**Table 3. Maximum Stress and Strain Values by Implant Design and Diameter in Bone Model B**

<table>
<thead>
<tr>
<th>Implant Diameter (mm)</th>
<th>1P Implants</th>
<th>2P Implants</th>
<th>Difference (1P vs. 2P)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Stress (Max.) (MPa)</td>
<td>Microstrain (Max.) (με)</td>
<td>Stress (Max.) (MPa)</td>
<td>Microstrain (Max.) (με)</td>
</tr>
<tr>
<td>3.0</td>
<td>54 (1P)</td>
<td>3637 (1P)</td>
<td>41 (2P)</td>
</tr>
<tr>
<td>3.7</td>
<td>47 (1P)</td>
<td>3143 (1P)</td>
<td>37 (2P)</td>
</tr>
<tr>
<td>4.1</td>
<td>30 (1P)</td>
<td>1980 (1P)</td>
<td>24 (2P)</td>
</tr>
<tr>
<td>6.0</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

* Refer Table 1.

1P indicates 1-piece; 2P, 2-piece.

---

**Table 4. Maximum Stress and Strain Values by Implant Design and Diameter in Bone Model C**

<table>
<thead>
<tr>
<th>Implant Diameter (mm)</th>
<th>1P Implants</th>
<th>2P Implants</th>
<th>Difference (1P vs. 2P)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Stress (Max.) (MPa)</td>
<td>Microstrain (Max.) (με)</td>
<td>Stress (Max.) (MPa)</td>
<td>Microstrain (Max.) (με)</td>
</tr>
<tr>
<td>3.0</td>
<td>77 (1P)</td>
<td>5438 (1P)</td>
<td>55 (2P)</td>
</tr>
<tr>
<td>3.7</td>
<td>53 (1P)</td>
<td>3633 (1P)</td>
<td>52 (2P)</td>
</tr>
<tr>
<td>4.1</td>
<td>41 (1P)</td>
<td>2704 (1P)</td>
<td>41 (2P)</td>
</tr>
<tr>
<td>6.0</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

* Refer Table 1.

1P indicates 1-piece; 2P, 2-piece.
vertical and lateral load stresses also decreased in inverse proportion to an increase in implant diameter, regardless of implant design. All stresses and strains observed under the prescribed boundary and load conditions were below approximately 54 MPa and 3637 με, respectively, in bone model B (Table 3), and 55 MPa and 3690 με, respectively, in bone model C (Table 4). The 12% exception to the latter finding, however, was exhibited by the 3.0-mm diameter 1P implant (smallest implant diameter in the study) placed into bone model C, which had minimal (1 mm) cortical bone thickness (Tables 1 and 4). In this model, stress values were higher than the bone yield point, which amounted to pathological loading. This was the worst outcome in Experiment 2.

In experiment 3, all stresses and strains observed under the boundary and load conditions for 1P and 2P implants subjected to 0-degree angle occlusal load vector were <29 MPa and 1929 με, respectively, for all implant diameters. These results do not account for the nonlinear response of bone. Comparison of 0- and 30-degree angle load directions indicated that off-axis loading generated significantly higher stress levels in bone than axial loading (Tables 5 and 6). For 1P and 2P implants subjected to 30-degree angle, occlusal load vector were higher up to 47 MPa and 2750 με, respectively. Data suggest that those stresses will increase proportionately to the distance of off-axis loading. Increasing bone quality equates with a greater ability to withstand higher occlusal load magnitudes without failure. Cortical bone has higher stiffness values than the trabecular cancellous bone. Although bone stress values increased in cortical bone compared with cancellous bone, the mechanical properties of cortical bone were better able to bear higher stresses than cancellous bone. In that way, higher density bone will allow for better stabilization of the implant assembly as compared with lower density bone.

In experiment 4, up to approximately 50% BIC did not affect maximum bone stress values, but stress levels began rising as the % BIC decreased to <50%. The reduced % BIC slightly increased local stresses around the implant threads. Highest stress contour bands were located in the crestal bone region, but progressively decreased deeper within the trabecular bone. For most evaluations, stress levels within trabecular bone were less than 5 MPa under the prescribed boundary conditions based on typical von-Mises stress contours.

**DISCUSSION**

Vertical and transverse occlusal loads produce stress gradients in implant systems and surrounding bone. The manner in which an implant system transfers the resulting axial and off-axial forces and bending moments to the supporting bone directly affects its survival and long-term crestal bone maintenance. Many different variables can influence load distribution, such as implant geometry and dimensions, material properties, surface characteristics, the % BIC, bone volume and properties, prosthesis type, and load vector angles relative to the implant axis. FEA has been widely used since the late 1970s to convert these complex biomechanical relationships into more simplistic mesh elements, analyze them at a functional level, and then resemble the data to predict the patterns and effects of stress on dental implants and the surrounding tissues. This task requires the formulation of many assumptions and theoretical constructs, all of which entail some inherent margin of error.

In this study, each FEA study model consisted of a simulated implant embedded in a block of bone measuring 1 to 5 mm in thickness around the implant. Although FEA implant studies commonly use this type of model, the design inherently skews generated stress values because of the limited bone volume. An implant placed into a rectangular bone block with 1 to 5 mm of bone thickness on the simulated buccal and lingual aspects of the implant, but significantly greater bone volume on its simulated mesial and distal aspects, may more accurately simulate stress dispersion in bone comparable with that of an actual implant placed into the alveolar process.
vated stress values in this study, the use of a rectangular bone block would probably not have altered the observed relationships between study variables. In other words, reducing simulated lingual and buccal plate thickness of an implant placed in a rectangular bone block model would still result in elevated bone stress values, but those stress values would probably not be as high as stresses generated in a cylindrical bone block model with less volume to absorb stresses.

Other discrepancies in this study included the assumed 100% bonding between the implant and abutment in the 2P model, and the assumed 100% BIC in some experiments. In reality, manufacturing tolerances would not allow a 100% bond between all interlocking implant-and-abutment geometrical surfaces in this design, but the friction-fit interface that is achievable would probably render the difference between the simulated model and the actual components negligible in terms of stress generation. In a similar manner, performing the experiments in question with <100% BIC might affect recorded stress levels somewhat, but the relationships between the variables would probably not be significantly different. These preliminary findings on the differences in load distribution between 1P and 2P implant designs require further investigation.

**CONCLUSIONS**

Within limitations of this study, 1P and 2P implant models of equivalent endosseous geometries and diameters did not exhibit significantly different stress distributions in bone. According to these analyses, wider implant diameters, thicker periimplant bone, thicker cortical bone, reduced load magnitudes, and smaller load vector angles to the long axis of the implant can help to reduce bone stress levels.

**DISCLOSURE**

The authors claim to have received the numeric modules of implant from Zimmer.

**ACKNOWLEDGMENTS**

The authors thank Mike Werner, Zimmer Dental, Carlsbad, CA, for his technical support.

**REFERENCES**


---

**Table 6. Maximum Stress and Strain Values by Implant Diameter and Design in Bone Model B*: Comparison of Load Direction (0-Degree Angle vs. 30-Degree Angle)**

<table>
<thead>
<tr>
<th>Implant Diameter (mm)</th>
<th>Unit</th>
<th>1P Implants</th>
<th>2P Implants</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td>30 degrees</td>
<td>0 degrees</td>
</tr>
<tr>
<td></td>
<td></td>
<td>30 degrees</td>
<td>0 degrees</td>
</tr>
<tr>
<td>3.0 Stress (MPa)</td>
<td>54</td>
<td>29</td>
<td>(–) 46</td>
</tr>
<tr>
<td>Microstrain (με)</td>
<td>3637</td>
<td>1929</td>
<td>(–) 47</td>
</tr>
<tr>
<td>3.7 Stress (MPa)</td>
<td>47</td>
<td>27</td>
<td>(–) 43</td>
</tr>
<tr>
<td>Microstrain (με)</td>
<td>1920</td>
<td>1576</td>
<td>(–) 20</td>
</tr>
<tr>
<td>4.1 Stress (MPa)</td>
<td>–</td>
<td>–</td>
<td>–</td>
</tr>
<tr>
<td>Microstrain (με)</td>
<td>–</td>
<td>–</td>
<td>–</td>
</tr>
<tr>
<td>4.7 Stress (MPa)</td>
<td>30</td>
<td>16</td>
<td>(–) 47</td>
</tr>
<tr>
<td>Microstrain (με)</td>
<td>1980</td>
<td>1576</td>
<td>(–) 20</td>
</tr>
<tr>
<td>6.0 Stress (MPa)</td>
<td>–</td>
<td>–</td>
<td>–</td>
</tr>
</tbody>
</table>

* Refer Table 1.

1P indicates 1-piece; 2P, 2-piece.


