

# EVALUATING PARAMETERS OF OSSEOINTEGRATED DENTAL IMPLANTS USING FINITE ELEMENT ANALYSIS—A TWO-DIMENSIONAL COMPARATIVE STUDY EXAMINING THE EFFECTS OF IMPLANT DIAMETER, IMPLANT SHAPE, AND LOAD DIRECTION

Eric P. Holmgren, MS  
Robert J. Seckinger, DMD, MDS  
Leslie M. Kilgren, PhD  
Francis Mante, DMD, PhD

## KEY WORDS

Finite element analysis  
Dental implants  
Stress analysis  
Stress distribution  
Bite force direction

*This manuscript represents the research endeavors of Eric Holmgren, a student at the University of Pennsylvania School of Dental Medicine, along with three of his senior colleagues. It was the winner of the IRG/IADR Young Investigator Award given by the American Academy of Implant Dentistry.*

*The authors are at the University of Pennsylvania School of Dental Medicine, Philadelphia, Pa.*

Finite element analysis (FEA) has been proven to be a precise and applicable method for evaluating dental implant systems. By means of FEA, a parasagittal model was digitized from a computed tomography (CT)-generated patient data set, and various single-tooth, osseointegrated, two-dimensional dental implant models were simulated. The specific aims of the study were to: (1) examine the effect of implant diameter variation (3.8 mm–6.5 mm) of both a press-fit, stepped cylindrical implant type and a press-fit, straight cylindrical implant type as osseointegrated in the posterior mandible; (2) compare the stress-dissipating characteristics of the stepped implant versus the straight implant design; and (3) analyze the significance of bite force direction (vertical, horizontal, and oblique 45°) on both implant types. The results of the FEA suggested that (1) using the widest diameter implant is not necessarily the best choice when considering stress distribution to surrounding bone, but within certain morphological limits, for both implant types, an optimum dental implant exists for decreasing the stress magnitudes at the bone-implant interface; (2) stress is more evenly dissipated throughout the stepped cylindrical implant when compared to the straight implant type; and (3) it is important in FEA of dental implants to consider not only axial forces (vertical loading) and horizontal forces (moment-causing loads), but also to consider a combined load (oblique bite force), since these are more realistic bite directions and for a given force will cause the highest localized stress in cortical bone. The theoretical analysis performed implies that clinically, whenever possible, an optimum, not necessarily larger, dental implant should be used based on the specific morphological limitations of the mandible and that a stepped cylindrical design for press-fit situations is most desirable from the standpoint of stress distribution to surrounding bone.

## INTRODUCTION

**D**ental implants are a very popular alternative to dentures. Implant research and clinical applications have become very sophisticated in both orthopaedic and dental fields, implying that use of an accurate theoretical tool is imperative. One such tool is finite element analysis (FEA), which has been proven to be a precise and applicable method for evaluating dental implant systems.<sup>1,2</sup>

In general, dental implants restore the function of missing or removed teeth. They are anchored in the underlying alveolar bone while protruding through the socket into the oral cavity so as to provide abutment posts for single-tooth, fixed bridge, or full arch appliances.<sup>3</sup> Dental implants are unique in this sense since they are transgingival, implying that they undergo occlusal forces of various magnitudes and directions, some of which can be very large. Thus, the structural integrity and placement of an implant are crucial for implant success. The use of FEA to evaluate the structural mechanics of dental implants is ideal because it allows one to evaluate various parameters such as implant diameter, implant shape, and load direction. All of these parameters play large roles in evaluating the efficacy of the structural mechanics of dental implants and thus, the overall success of the dental implants.

## BACKGROUND

FEA has been widely used in the literature to evaluate dental implant design and function<sup>4-12</sup> and has been proven to be a good theoretical tool for analyzing dental implant systems.<sup>1,2</sup> To date, FEA has been lacking in the area of testing the effects of implant diameter on stress distribution in a compound bone model as a function of load direction. One FEA model<sup>9</sup> has examined implant diameter variation, but the model used a less accurate tri-

angular element type—it considered only one type of implant (press fit, cylindrical), and an oblique bite force direction was not investigated. The previous literature mentioned above has shown that the cortical bone-implant interface typically has a high stress concentration. The analyses in this study mostly concentrate near the cortical bone-implant interface, since high stresses in this area may lead to unwanted bone resorption<sup>12</sup> and thus decreased likelihood of implant success.

## STUDY AIM

The initial stages of the study involved creating a solid model of a parasagittal section of a mandible digitized from multiplanar reformatting of a computed tomography (CT)-generated patient data set. The results were then displayed as stress contours by material type (implant, cortical bone, and trabecular bone). The specific aim was to: (1) examine the effects of implant diameter variation (3.8 mm, 4.5 mm, 5.5 mm, 6.5 mm) of both a stepped cylindrical and straight cylindrical implant on a compound bone model of the posterior mandibular region. Of specific interest was the subcrestal region of the cortical bone-implant interface; (2) analyze the stress contours and stress distribution of a stepped cylindrical implant design as compared to a straight cylindrical implant design; and (3) examine the effects of bite force direction (vertical, horizontal, oblique 45°) on both implant designs.

## FEA—THE CONCEPT

For problems involving complicated geometries, it is very difficult to achieve an analytical mathematical solution. Therefore, the use of a numerical method such as the finite element method is required. The finite element method is a technique for obtaining a solution to a complex problem by subdividing the problem into a collection of smaller and simpler problems that can be solved using numerical techniques. An overall approximate solution to the original problem is deter-

mined based on the combined solution from the smaller, simpler subproblems. In other words, FEA is a method whereby, instead of solving the problem for the entire body in one operation, one formulates the equations for each finite element and combines them to obtain the solution to the whole body.<sup>3,13</sup>

FEA was initially developed in the early 1960s to solve structural problems in the aerospace industry but has been extended to solve problems in the heat transfer, fluid flow, mass transport, and electromagnetic realm. Instead of solving FEA problems with large supercomputers that took days to process, FEA problems can now be solved using a personal computer (PC) in a matter of seconds, due to the vast increase in computer technology.

Since the structural integrity of the dental implant-bone system is so vital to implant success, use of FEA is a superb tool. It is also an aesthetic process in that the mathematical model one develops can be displayed graphically to resemble the actual structure being modeled. The finite element organization resembling the model is called the "finite element mesh." The process of creating the mesh, elements, their respective nodes (point boundaries surrounding the element), and defining boundary conditions (points that fix the model in space and that oppose the external force) is termed "discretization" of the model.

## FEA—ASSUMPTIONS

As with all theoretical analyses, there are certain assumptions that need to be made to make the modeling and solving process possible. The trabecular bone network will be ignored simply because the capability to determine the trabecular pattern is not available. Therefore, it will be assumed that trabecular bone has a solid pattern inside the inner cortical bone shell. Both bone types will be modeled to be linear, homogeneous, and isotropic with respect to their geometrical boundaries. In other words, the bone material properties



FIGURE 1. Section of posterior mandible from actual patient's CT data.

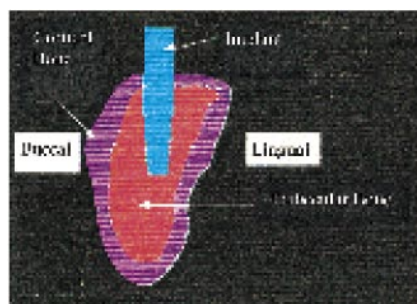


FIGURE 2. 2-D model before meshing. Parasaagittal view of second molar region with stepped cylindrical implant inserted with abutment. Cortical bone geometry was digitized from CT scan data of a patient's mandible.

are assumed to be the same in every plane and do not exhibit any nonlinear stress-strain characteristics or plasticity. The FEA model will also assume a state of osseointegration, meaning that cortical and trabecular bone will be bonded to the implant. Since no exact measurements of mandibular cortical and trabecular bone material proper-

ties (Young's modulus and Poisson's ratio) have been reported, material properties have been extrapolated from an exhaustive list of femoral bone properties.<sup>3</sup>

Whether to perform a two-dimensional (2-D) or three-dimensional (3-D) study is an important question in FEA. It is usually suggested that, when comparing the qualitative results of one case with respect to another, a 2-D model is efficient and just as accurate as a 3-D model.<sup>3</sup> Although the time needed to generate FEA models is decreasing with increasing computer technology, there is still a justified time and cost savings when using a 2-D model over three dimensions when it is appropriate.

The boundary conditions to be selected for the model are of the support type as opposed to a suspension type. Suspension-type constraints require that all of the exterior nodes on the cut ends of the bone segment be con-

strained. This will not be used for the 2-D models since it requires a long, 3-D bone segment. Support-type constraints are applied near the base of the bone segment in position so as to oppose the external load of the model in space (*ie* there is no free body motion). This type of constraint is ideal for 2-D studies, since it might skew results near the base of the model but not interfere with the apex of the model near the cortical bone-implant interface where the main interest lies.

For 2-D analysis, assuming either a plane stress, plane strain, or axisymmetric type of model is required. Plane stress analysis infers that the structure to be modeled will not have any forces (or stresses) out of its plane but that it can have deformations (or strains) out of its plane. In other words, if modeling in the *x-y* plane, no stresses will occur in the *z*-plane but deformations could occur in the *z*-plane. Plane stress analysis is used for structures such as flat plates that are subjected to loading within the plane.

This analysis is also used for thin disks under pressure or centrifugal loading. The opposite is true for plane strain analysis, which assumes that no deformation will occur in the *z*-direction but that stress can occur in the *z*-direction. Plane strain analysis is used for structures in which one dimension is much larger than the other two dimensions, and the cross-section of interest is perpendicular to the long axis.<sup>14</sup> This type of analysis is best for a model of the human mandible. An axisymmetric model is appropriate for structures that have symmetry about the loaded axis, such as symmetrical implants in cylindrical bone cores.<sup>15</sup> This is not suitable for analyzing the specific mandibular models in this study since they are nonsymmetrical. The type of analysis needed is mainly driven by which type of element should be used for the model. For example, a shell-type element works best for a plane stress model and a solid-type element works best for a plain strain element.<sup>14</sup>

Material	Young's Modulus	Poisson's Ratio
Ti-6AL-4V <sup>16</sup>	$1.1 \times 10^5$ mPa	0.33
Cortical bone <sup>3</sup>	$1.5 \times 10^4$ mPa	0.3
Trabecular bone <sup>3</sup>	$1.5 \times 10^2$ mPa	0.3

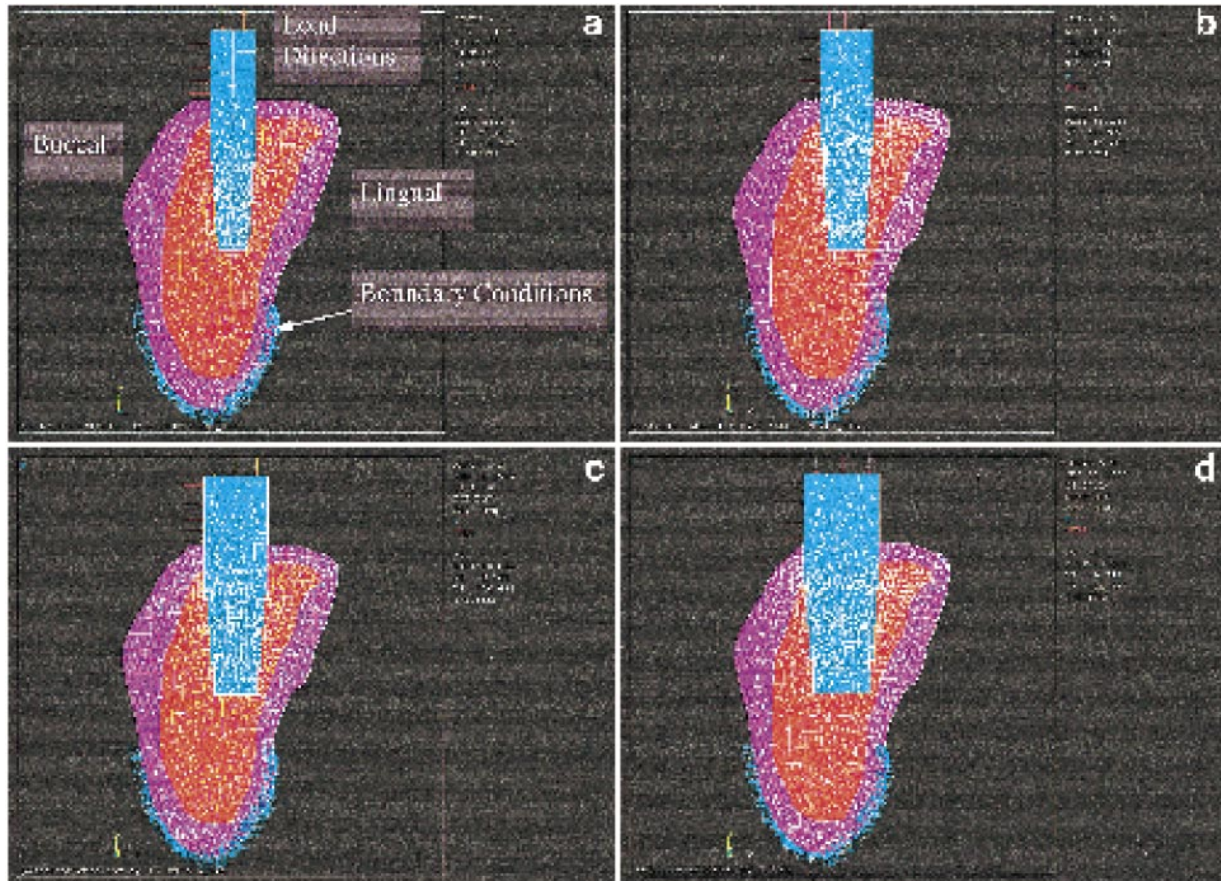


FIGURE 3. (a) 2-D model with 3.8-mm-diameter implant as placed in the mandible. All models are meshed and display element configurations. Support-type boundary conditions are shown. Oblique force equivalent is also displayed as the load direction. (b) Model with 4.5-mm-diameter implant. (c) Model with 5.5-mm-diameter implant. (d) Model with 6.5-mm-diameter implant.

Although many assumptions are required to undertake a finite element analysis, FEA has been proven to be an effective tool for evaluating dental implant systems.<sup>3</sup>

#### FEA—THE METHOD

##### *Preprocessing*

The solid modeling and FEA were performed on a G5-200 PC with 200 MHz Pentium technology (Gateway 2000, North Sioux City, S Dak) using ANSYS University Edition Software, version 5.3 (SASIP, Inc). As seen in Fig 1, patient-specific CT data can be digitized using ANSYS. In doing so, keypoints (selected points along the inner and outer cortical bone boundaries) from the CT data were identified and used to generate a parametric representation of the outer and inner contours of the observed cortical bone. The compound

bone model that was digitized and used throughout the study was generated from structure taken from the posterior molar region. Trabecular bone was assumed to fill the space inside the cortical bone inner boundary. This parametric representation was then transferred to the computer, and the simulated implant was inserted as seen in Fig 2.

Implant geometries were parameterized in a similar fashion from specifications for the stepped cylindrical implant (Frialit-2® dental implant, Fria-tec, Germany). In all cases, the implants were 13 mm in length and were assumed to have a 6-mm titanium abutment attached, except for the 6.5-mm-diameter implant, where a 10-mm length was used.

ANSYS requires only Young's Modulus (elastic modulus) and Poisson's ra-

tion as material input for structural analysis. Table 1 lists the material properties used in the model.

As mentioned previously, since the plane strain type of analysis is best suited for this study, use of an eight-noded solid quadrilateral element will give best results<sup>3</sup> (as opposed to a triangular solid element used in other studies<sup>9</sup>). After the element type was chosen, the material properties established, and the solid model complete, the next step was to create the finite element mesh. The first stage in meshing involved making sure the proper meshing attributes (element type and material properties) were assigned to the proper area (cortical bone, trabecular bone, and implant). Then, the correct meshing size (ranging from coarse to fine) was established using the ANSYS "smart-size" default option, which

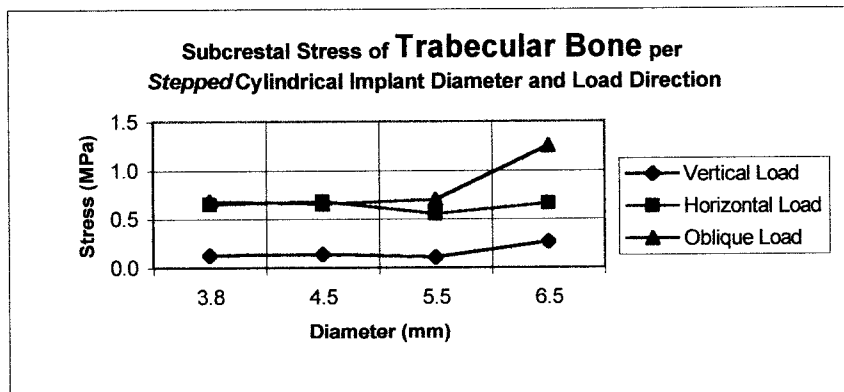
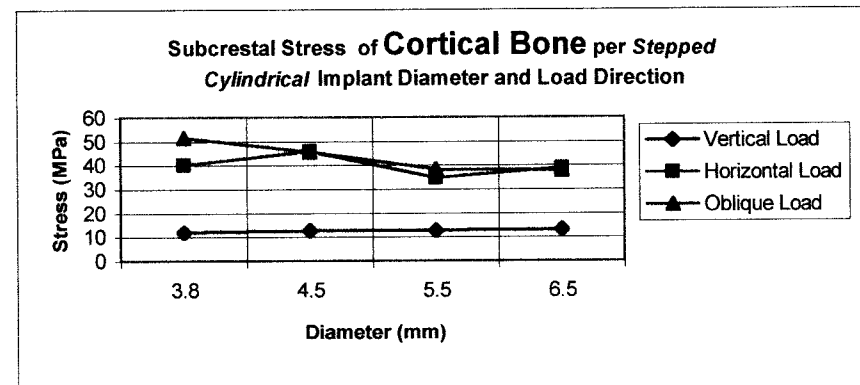
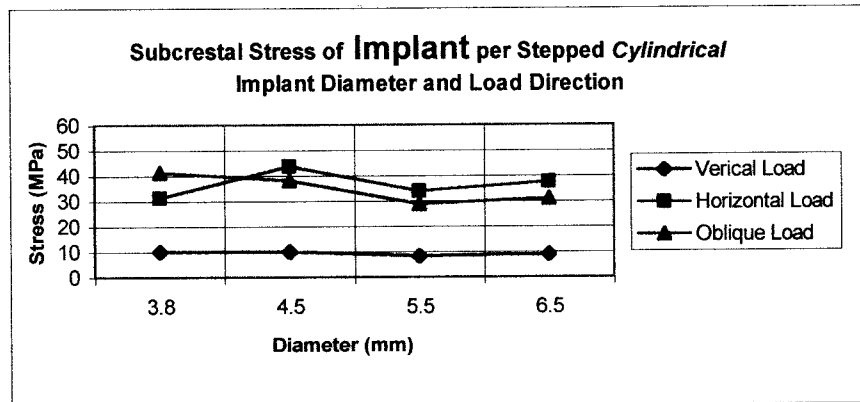


FIGURE 4. Subcrestal stress of (a) implant, (b) cortical bone, and (c) trabecular bone per stepped cylindrical implant diameter and load direction.

optimizes the appropriate coarseness for the mesh based on the geometrical boundaries of the model.<sup>14</sup> Lastly, the actual meshing operation was undertaken to break the solid model into connected elements. The results of these operations can be seen in Fig 3. The number of nodes and elements for the models in this study averaged a total of 2800 and 900, respectively.

The loading and boundary conditions are characterized in Fig 3a. The loading (bite force) used for the study was equivalent to 285 psi ( $\approx 2$  mPa), which is comparable to a 5-lb (or 100 N) force on a 3.8-mm-diameter implant. The load magnitudes were chosen to resemble related literature<sup>9</sup> and were applied as surface loads directly to the 6-mm abutment. For direct com-

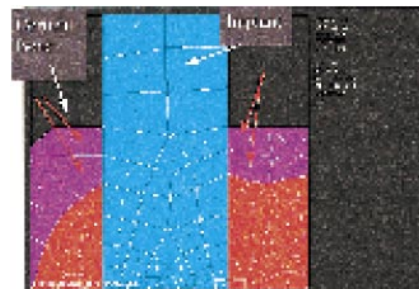


FIGURE 5. Enlarged picture of cortical bone-implant interface. Red arrows point to cortical bone elements of interest.

parison between horizontal and vertical loading, the same magnitude was used for both directions. For an oblique load of 45°, both horizontal and vertical load components were applied to the 6-mm abutment. Boundary conditions as mentioned previously were of the support type and were applied at nodes at the base of the mandibular model. The major area of interest is near the bone-implant interface, and interference caused by these boundary constraints was not significant there.

#### Solution phase

During this phase of analysis, the meshed model was solved as a function of the applied load step (bite force magnitude and direction). A common finite element approach for structural mechanics problems is use of the displacement method. The displacement method determines the displacement of an element boundary by solving for displacement of the nodes surrounding an element as a function of an external load, that is, a bite force. Since ANSYS requires only Young's Modulus (ratio of stress to strain) and Poisson's ratio (ratio of strain in an  $x$ -direction to strain in a  $y$ -direction for a 2-D analysis in the  $x$ - $y$  plane) as material properties, the additional unknowns, strain and stress, can be determined within an element as a function of nodal displacement. Overall, during this phase of analysis, the meshed model with boundary conditions can be solved as a function of the current load step—either a vertical, horizontal, or oblique (45°) load direction for a given implant

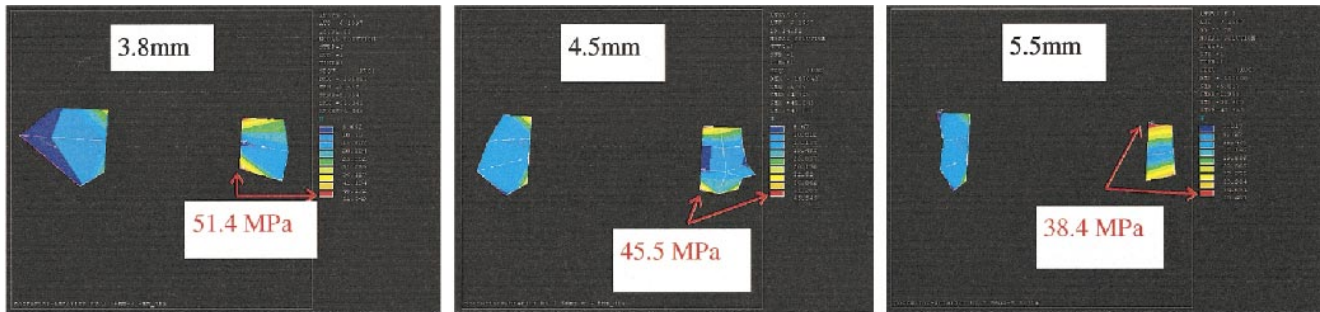


FIGURE 6. Cortical one neck elements. Stress comparison per stepped implant diameter for an oblique load. The maximum stress magnitudes in red decrease with increasing implant diameter up to 5.5 mm.

type and diameter. What is being solved for, in general terms, are the nodal displacements of an element as a function of the load (bite force). Subsequently, the strains and stresses distributed from element to element can be solved for and then displayed in the postprocessing phase.

#### Postprocessing Phase

Results were graphically displayed as stress contours, specifically as Von Mises equivalent stresses:

$$\sigma_e = [\sigma_1^2 - \sigma_1\sigma_2 + \sigma_2^2]^{1/2}$$

which is a function of the principal stress within an element:

$$\sigma_1, \sigma_2 = 0.5 [(\sigma_x + \sigma_y) \pm \sqrt{(\sigma_x - \sigma_y)^2 + 4\tau_{xy}^2}]^{1/2}$$

where

$\sigma_x$  = lateral stress perpendicular to

the long axis of the implant,

$\sigma_y$  = longitudinal stress directed

along the long axis of the

implant, and

$\tau_{xy}$  = shear stress.

ANSYS calculates nodal data by integrating over a region surrounding each node. If a node is shared by elements of differing material properties, such integration is likely to produce inaccurate results at that node. Therefore, stresses were graphically displayed in only one material (implant, cortical bone, or trabecular bone) at a time, and not across material boundaries.

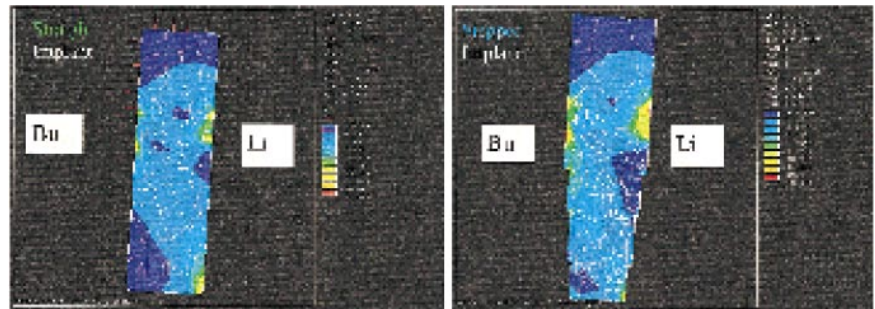


FIGURE 7. Oblique load implant type comparison. Stress contours of the straight implant versus stepped implant. Implant comparison with both implants at 5.5-mm diameter and both with oblique load (not shown with stepped implant). Bu = buccal, Li = lingual.

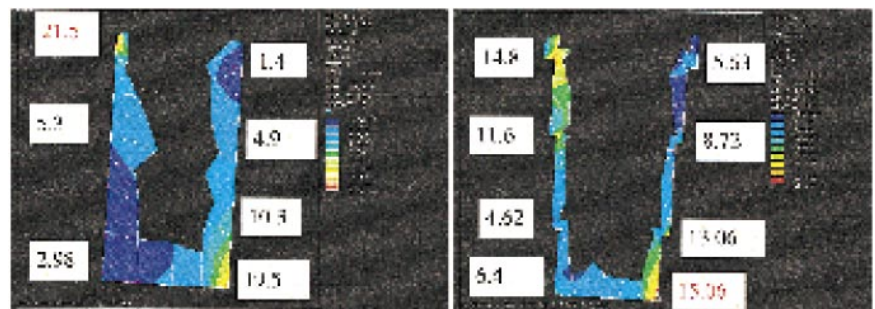


FIGURE 8. Oblique load elemental and "notch" stress distribution. Left picture is straight implant and right is stepped implant. The stress magnitude values are more evenly distributed along the stepped implant interface.

## RESULTS AND DISCUSSION

### Stress magnitudes as a function of implant diameter variation

Figure 4 graphs the maximum Von Mises stress magnitudes within the stepped implant (Fig 4a), the cortical bone (Fig 4b), and the trabecular bone (Fig 4c), all as a function of load direction and implant diameter. The maximum stress values were measured in

the cortical bone-implant neck elements as seen in Fig 5. The stress trends for the straight implant are similar to that of the stepped implant. The general trends for the figures are that as the implant diameter increases, the stress magnitude decreases. This is highly significant for the cortical bone (Fig 4b), since increasing the implant diameter can decrease the amount of stress imparted to the surrounding cortical bone neck elements by the im-

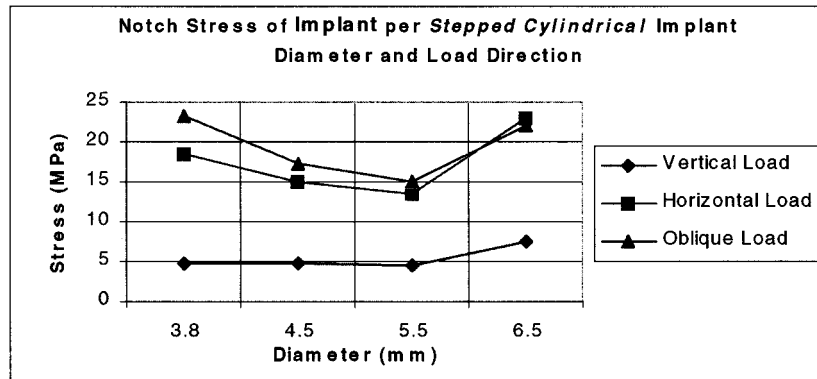


FIGURE 9. Notch stress of implant per stepped cylindrical implant diameter and load direction.

plant, thereby increasing the likelihood of implant success by decreasing unwanted cortical bone resorption.

However, Fig 4c shows that the maximum stress within the trabecular bone is higher for the 6.5-mm-diameter implant when compared to that of the other implant. Also, the stress within the implant and the cortical bone starts to level out as the implant diameter is increased beyond 5.5 mm. This trend implies that the morphological shape of the bone in this area of the mandible cannot accommodate the 6.5-mm-diameter implant. The 6.5-mm-diameter implant is starting to infringe on the amount of available space within this posterior mandibular section and would therefore be an inappropriate-sized implant for this patient's posterior mandible. This observation indicates that an optimum implant diameter exists for this patient that is less than 6.5 mm. If one is concerned with low cortical bone-implant interface stresses, then the optimum diameter would be 5.5 mm, since as in Fig 4b, this diameter transfers the lowest stress when a horizontal or oblique bite force is applied.

Stress contours for cortical bone elements nearest the cortical bone-implant interface can be displayed for further verification that increasing implant diameter, within morphological limits, decreases stress magnitudes at the cortical bone-implant interface. Figure 6 shows for an oblique load (most

severe loading situation) how the stress contours behave as a stepped implant diameter is varied (imagine the implant to be between the left and right selected elements). The red arrows show areas of maximum stress within the elements. Again, by selecting these cortical bone elements and solving, a stress distribution was scaled for more precise determination of stress magnitude, better allowing for direct comparison between implant designs.

#### *Implant stress contours of stepped versus straight implant design*

Figure 7 compares the stress distribution for stepped versus straight implants along the long axis of the implant. One can see that stress is dissipated more evenly along the stepped implant. Figure 8 indeed shows this trend better. By selecting elements along the implant and measuring their stress magnitudes, one can see that for the same extreme load condition (oblique load), the stepped implant type undergoes less of a dramatic stress magnitude change along the long axis of the implant when compared to a straight implant type. This is advantageous because it offers better structural stability and it transfers stress more evenly to surrounding bone, thus preventing regions of high bone resorption next to regions of low bone activity.

Figure 9 compares the maximum stress magnitudes along the notches

(values measured as in Fig 8) of the stepped implant as implant diameter varies per load direction. Figure 9 also indicates that the 5.5-mm-diameter stepped implant is the optimum choice from the standpoint of low notch stress. For the oblique and horizontal load conditions (moment-causing loads), an inverse trend up to 5.5 mm can again be noticed, once again implying that increasing implant diameter, within morphological limits, can decrease the maximum stress values along the implant, thus decreasing the amount of stress transferred to the surrounding bone.

#### *Stress magnitudes as a function of bite force direction*

Figures 4 and 9 further show that when comparing bite force direction on the implant as a function of implant diameter, the vertical load does not vary significantly with implant diameter. In contrast, the moment-causing load components, that is, the horizontal and oblique loads, are responsible for the trend of decreasing stress magnitudes with increasing implant diameter. One can also notice that the oblique load in most cases (Figs 4, 9) creates a larger stress magnitude than that of the horizontal component, implying that studies examining simply a horizontal bite force alone are not sufficient.

Figure 10 compares the stress contours of a cortical bone result with a 5.5-mm-diameter stepped implant undergoing an oblique load to that of the same with a vertical load. The zoomed elemental stress ranges show that the stress magnitudes for the oblique load situation are much greater than that of the vertical load situation.

#### CONCLUSIONS

The results of the study suggest the following.

1. The larger diameter implant is not always the best choice for minimizing cortical bone-implant interface stress. The models in this study using FEA have shown that an optimum implant

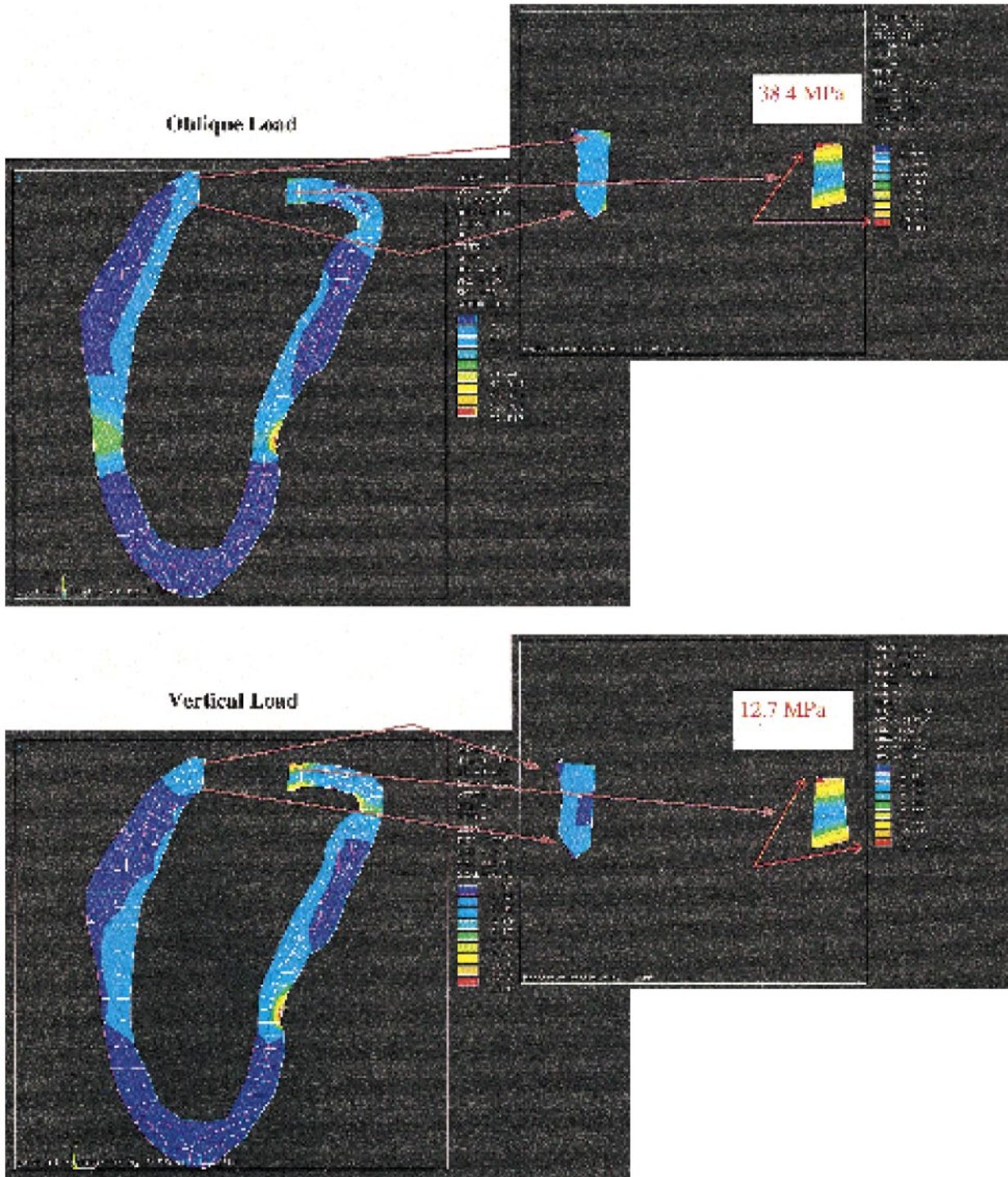


FIGURE 10. Both cortical bone solutions are of a stepped implant with a 5.5-mm diameter. Cortical neck elements are displayed to the right, and the burgundy lines designated the cortical neck elements origin. The oblique load causes over a threefold stress magnitude increase in surrounding cortical bone when compared to only a vertical load on the same implant. Maximum stresses are designated by red arrows.



diameter exists for this specific patient data set. The optimum choice usually correlates with being the largest implant diameter, within morphological limits, causing the least stress, when loaded, within the surrounding cortical bone while also causing minimal trabecular bone stress. Therefore, clinically, to increase implant-bone structural integrity and, thus, the useful life of the implant-based restoration, one should carefully consider the morphology of the bone involved and its subsequent ability to accommodate a chosen implant shape and size.

2. Stress is more evenly distributed throughout a stepped cylindrical implant when compared to a straight cylindrical implant; hence, the stepped implant is a better choice clinically for press-fit situations. Also, the notch stresses along the stepped implant are lowest, and thus optimum, with the 5.5-mm-diameter implant for this specific patient CT data.

3. It is important in the FEA of dental implants to consider axial forces (vertical load) and horizontal forces; it is also imperative to consider combined loads (oblique loads), since these are more realistic bite directions and, for a given force, will cause the highest localized stress in cortical bone.

#### REFERENCES

1. Koriath TWP, Dechow PC, Hanam AG. 3D finite element modeling and validation of a dentate human mandible. *J Dent Res.* 1992;71:781.
2. Baiamonte T, Abbate MF, Pizzarello F, Lozada J, James R. The experimental verification of the efficacy of finite element modeling to dental implant systems. *J Oral Implantol.* 1996;22:2.
3. Cowin SC. *Bone Mechanics.* Boca Raton, Fla: CRC Press; 1989.
4. Seckinger, RJ. A three-dimensional finite element stress analysis in and around the ISIS endosseous dental implant. MS thesis; Pittsburg, Penn: University of Pittsburg School of Dental Medicine; 1989.
5. Clelland NL, Ismail YH, Zaki HS, Pipko D. Three-dimensional finite element stress analysis in and around the screw-vent implant. *Int J Oral Maxillofacial Implants.* 1990;6:457-465.
6. Reiger MR, Fareed K, Adams WK, Tanquist RA. Bone stress distribution for three endosseous implants. *J Prosthet Dent.* 1989;61:223-228.
7. Reiger MR, Adams WK, Kinzel GL. A finite element survey of eleven endosseous implants. *J Prosthet Dent.* 1990;63:457-465.
8. Reiger MR, Mayberry M, Brose MO. Finite element analysis of six endosseous implants. *J Prosthet Dent.* 1990;16:6-11.
9. Matsushita Y, Kitoh M, Mizuta K, Ikeda H, Suetsugu T. The stress distribution of the hydroxyapatite implants: diameter effects on stress distribution. *J Oral Implantol.* 1988;16:6-11.
10. Kitoh M, Matsushita Y, Yamaue S, Ikeda H, Suetsugu T. The stress distribution of the hydroxyapatite implant under the vertical load by the two-dimensional finite element method. *J Oral Implantol.* 1988;14:64-71.
11. Bidez MW. An investigation into the effect of blade dental implant length of interfacial tissue stress profiles. In: Spilker RL, Simon BR, eds. *Computational Methods in Bioengineering.* New York: American Society of Mechanical Engineers; 1988.
12. Van Rossen IP, Braak LH, De Putter C, De Groot K. Stress-absorbing elements in dental implants. *J Prosthet Dent.* 1990;64:198-205.
13. Logan DL. *A First Course in the Finite Element Method.* Boston, Mass: PWS Publishing Co; 1986.
14. *Introduction to ANSYS*, release 5.3 course manual. Houston, Pa: SAS IP Inc.; 1996.
15. Holmes DC, Haganman CR, Aquilino SA, Diaz-Arnold AM, Stanford CM. Finite element stress analysis of IMZ abutment designs: development of a model. *J Prosthodontic.* 1997; 6:31-36.
16. Collings EW. *The Physical Metallurgy of Titanium Alloys.* Metals Park, Ohio: American Society for Metals; 1984. ■