FINITE ELEMENT ANALYSIS OF AN OSSEOINTEGRATED STEPPED SCREW DENTAL IMPLANT

J. P. Geng, BDS, DrPH, MScD
W. Xu, BEng, PhD
K. B. C. Tan, BDS, MSD
G. R. Liu, PhD

KEY WORDS

Dental implant
Finite element analysis
Stress distribution
Stepped screw implant
Cylindrical screw implant

An osseointegrated stepped screw dental implant was evaluated using 2-dimensional finite element analysis (FEA). The implant was modeled in a cross section of the posterior human mandible digitized from a computed tomography (CT) generated patient data set. A 15-mm regular platform (RP) Branemark implant with equivalent length and neck diameter was used as a control. The study was performed under a number of clinically relevant parameters: loading at the top of the transmucosal abutment in vertical, horizontal, and 45° oblique 3 orientations. Elastic moduli of the mandible varied from a normal cortical bone level (13.4 GPa) to a trabecular bone level (1.37 GPa). The study indicated that an oblique load and elastic moduli of the cortical bone are important parameters to the implant design optimization. Compared with the cylindrical screw implant, the maximum von Mises stress of the stepped screw implant model was 17.9% lower in the trabecular bone-implant area. The study also showed that the stepped screw implant is suitable for the cortical bone modulus from 10 to 13.4 GPa, which is not necessarily as strict as the Branemark implant, for which a minimum 13.4 GPa cortical bone modulus is recommended.

INTRODUCTION

Biomechanical optimization is an important objective in the design of dental implants. Although the success rates of some implant systems have been high, implant failures, even operated by a professional implantist, do occur.[1,2] The failure is in part due to the occlusal forces of various magnitudes and directions that the dental implants sustain, some of which can be very large.[3,4] One of the effective ways to maintain excellent clinical performance is to use a biomechanically optimized implant that provides a health stress-strain level required for normal bone resorption and deposition processes at the implant site. Most efforts have been directed at optimizing implant geometry

J. P. Geng, BDS, DrPH, MScD, *is with the Department of Dentistry, China Medical University Huayu Hospital, Shaoxing, 312030, ZJ, P. R. China.*

W. Xu, BEng, PhD *is with the Centre for Biomedical Engineering, School of Engineering, University of Surrey GU2 7XH, United Kingdom. Address correspondence to Dr Xu.*

K. B. C. Tan, BDS, PhD, *and* ***G. R. Liu, PhD*** *are with the National University of Singapore.*

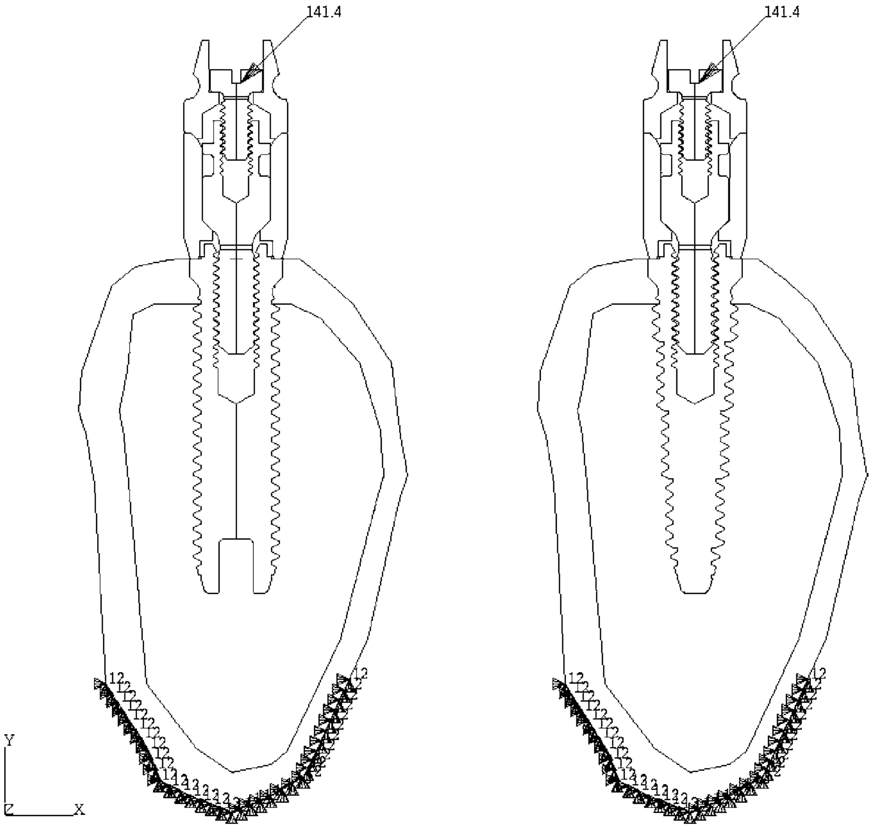

FIGURE 1. The 2 implant-bone systems: the cylindrical screw osseointegrated dental implant (left) and the stepped screw osseointegrated dental implant (right).

in order to maintain a beneficial stress level at the bone-implant interface. The effects of various parameters such as load direction and bone quality on the performance of a dental implant are very important.[5–9] A better understanding of these effects will lead to a significant improvement in the design of dental implants.

Finite element analysis (FEA) is an effective tool used to evaluate the biomechanical characteristics of different types of dental implants. The literature reflects that it has been widely used to model the design and functionality of dental implants and predict features of design optimization.[10] A key factor for the success or failure of a dental implant is the manner in which stresses are transferred to the surrounding bone.[11] Vertical and transverse loads from mastication induce axial forces and bending moments and result in stress gradients in the implant as well as in the bone.[5] Forces and moments transferred from implants to the surrounding bone depend on the type of loading, the bone-implant interface, implant geometry, the prosthesis type, and the quantity and quality of the surrounding bone.[10] Researchers can predict stress distributions in the contact area of an implant in cortical bone and around the apex of an implant in trabecular bone with FEA.

Implant geometry includes length, diameter, and shape. The optimum length and diameter necessary for long-term success depends on the bone support condition. If the bone is in a normal condition, length and diameter are not significant factors for implant success. However, if the bone condition is poor, large-diameter implants are recommended, and short implants should be avoided.[2,12–14]

With regard to implant shape, previous theoretical analyses[2] imply that clinically, whenever possible, an optimum but not necessarily a larger dental implant should be used based on the specific morphologic limitations of the mandible.

Holmgren et al[2] reported that a stepped cylindrical design for press-fit situations is most desirable, as the stress distribution in the surrounding bone becomes more uniform. Using FEA to analyze a parasagittal model digitized from a computed tomography (CT) generated patient data set, various single-tooth implants were simulated using 2-dimensional osseointegrated dental implant models. The results suggested that the stress was more evenly dissipated throughout the stepped cylindrical implant compared with the straight implant type. However, Mailath et al[12] also used FEA to compare cylindrical to conical implant shapes when exposed to physiologic stresses, and they examined the occurrence of stress concentrations at the site of implant entry into the bone. They reported that cylindrical implants were preferable to the conical implant shapes. Siegele and Soltesz[15] compared cylindrical, conical, stepped, screw, and hollow cylindrical implant shapes using FEA. Both a fixed bond (simulating complete load transfer with bioactive materials) and a pure contact (only compression transfer with bioinert materials) without friction between implant and bone were considered as interface conditions. The results demonstrate that different implant shapes lead to significant variations in stress distributions in the bone. The authors stated that implant surfaces with very small radii of curvature (conical) or geometric discontinuities (stepped) induced significantly higher stresses than smoother shapes (cylindrical, screw-shaped).

Although the biological bone remodeling process at the implant site cannot be confirmed on the basis of FEA alone, the stress-strain level can be analyzed to provide a valuable reference for the design optimization of a dental implant. The main purpose of this study was to use FEA to evaluate the osseointegrated stepped screw dental implant in terms of stress distribution and maximum stress level, as both could affect the bone remodeling at the implant site, and hence implant survival. Because not all implant geometry and input loads to the bone-implant system are crucial factors in sustaining a dental implant, the secondary purpose of this study was to investigate the significance of clinical relevant parameters to the design optimization of dental implants.

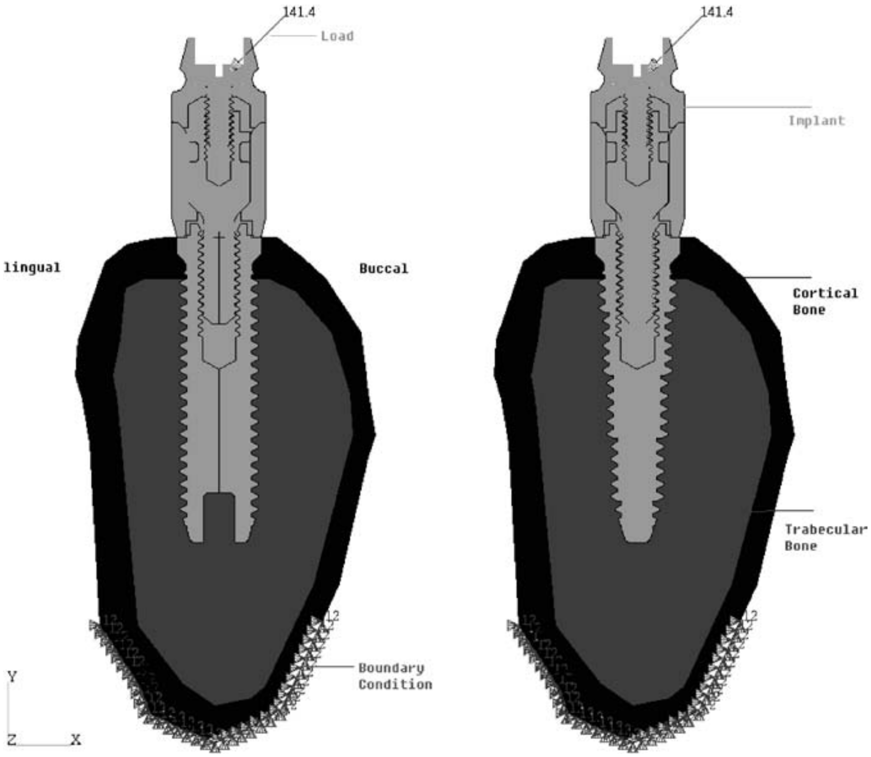

FIGURE 2. The finite element analysis (FEA) load and boundary conditions.

MATERIALS AND METHODS

An osseointegrated 5-stepped screw dental implant, 15 mm in length with a neck diameter of 4.1 mm, was modeled using 2-dimensional FEA. Cortical and cancellous bone were also modeled to represent the standard B-L cross section of the posterior human mandible digitized from a CT-generated patient data set. The thickness of cortical bone around the implant neck was set at 2 mm. A cylindrical screw implant (Branemark implant, Nobel Biocare, Sweden) of equivalent length and neck diameter, with similar thread and pitch, was used as a control.

FEA assumptions

A 2-dimensional finite element model of an implant-bone system was developed using MSC/PATRAN 8.5 (MSC Software Corp, Santa Ana, Calif; Figure 1). The use of the 2-dimensional model was based on the fact that a proper 2-dimensional model is much more efficient compared with its 3-dimensional counterpart, and the results can be as accurate if only a qualitative study is performed.[16]

Plane strain analysis was used for structures in which 1 dimension was much larger than the other 2 dimensions and the cross section of interest was perpendicular to the long axis. This type of analysis is best for the human mandibular model. In this analysis, therefore, a plane strain assumption was adopted, and a representative B-L cross section

TABLE 1

Material properties of implant-bone system

	Young's Modulus (GPa)	Poisson's Ratio	Reference
Implant (CPT)	117	0.30	21
Cortical bone	13.4	0.30	17
Trabecular bone	1.37	0.31	18

TABLE 2

Conditions of cortical bone ranging between normal and weak bone

Condition	Young's Modulus (GPa)	Poisson's Ratio	Reference
1	13 400	0.30	Normal condition
2	10 000	0.30	
3	7500	0.30	
4	5000	0.30	
5	1370	0.31	Trabecular bone level

of the posterior human mandible was modeled. However, the implant thickness in this study was defined as 4 mm to avoid potential inaccuracy due to B-L horizontal and oblique load.

Boundary conditions

The nodes over the free edges of cortical bone were constrained in the *x*-, *y*-, and *z*-directional rigid movement. The boundary conditions of 2 implant-bone systems are shown in Figure 2. In this study, it was assumed that there was no-slip, 100% rigid interfacial contact between bone and implant.

Material properties

The material properties of the cortical and trabecular bone and implant are shown in Table 1. As the other physiological properties do not have a major contribution to a linear FEA analysis, the corresponding cortical bone qualities were represented by their elastic moduli (Table 2).

Finite element model

The finite element model was created using Element Topology: Quad4. In order to reduce the computation time of FEA, a pilot study was carried out by the authors to determine the adequate global edge length of the elements at different locations. The overall global edge length of the model was 0.5 mm; however, the global edge length at the interface between implant and bone was about 0.2 mm.

The control and stepped screw implant models were compared under different conditions. Different loadings in a normal cortical bone condition were compared when a nodal force was applied on the top of the transmucosal abutment, and 3 cases were considered: (1) vertical load of 100 N, (2) horizontal load of 100 N, and (3) combined oblique load of 141 N at a 45° inclination. Different cortical bone properties in an oblique load condition were then compared with the normal cortical bone condition. For bone-implant finite element modeling, the essential property of the bone is an elastic modulus. Clinically, the elastic moduli of cortical bone varies from a normal bone (13.4 GPa) to a weak bone (1.37 GPa, similar to the trabecular bone). In this study, 5 different elastic moduli of cortical bone ranging between 13.4 and 1.37 GPa were modeled (Table 2). The values of 13.4 and 1.37 GPa, respectively, for cortical and trabecular bone were taken from the literature.[17,18]

As the von Mises stress has been commonly used to characterize FEA studies[2,15,19] and provides a convenient representation of the stress situation, in this study von Mises stress distributions adjacent to the bone-implant interface were investigated for both stepped and regular platform (RP) Branemark implant models.

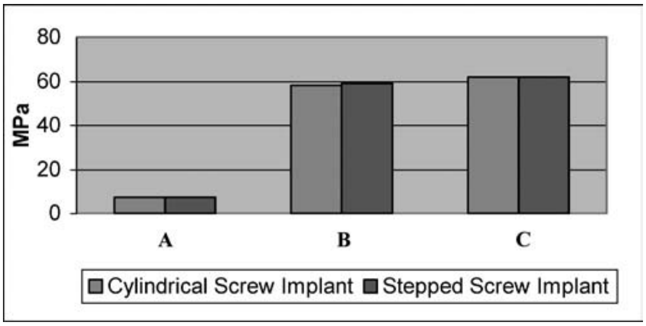

FIGURE 3. The maximum von Mises stress in the cortical bone adjacent to the implant neck under 3 different loads (condition A, vertical load, 100 N; condition B, horizontal load, 100 N; and condition C, oblique load, 141 N at 45° inclination).

RESULTS

The main purpose of this work was to simulate the stress distribution and maximum stress level for evaluating the sustainability and optimization of the implant system. Both stress distributions and the maximum stresses presented in this section are average von Mises element stresses. The results reveal the effects of different loadings and properties of the bone to the stress distribution and the maximum von Mises stress in the cortical and trabecular bone adjacent to the implant.

Different load conditions in normal cortical bone

Comparing stepped screw implants with the cylindrical screw implant models using similar implant length and neck diameter resulted in the following:

- In cortical bone immediately adjacent to the implant neck, there were no significant differences in stress distribution under different load conditions. For all loading conditions, both models have approximately equivalent maximum stress in the cortical bone (Figure 3).
- Figure 4 shows the corresponding L1 to B10 locations between the stepped screw and cylindrical screw implant along the trabecular bone-implant interface. In the trabecular bone adjacent to the implant for the stepped screw implant model, stresses were more uniformly distributed under an oblique loading condition (Figure 5). Under oblique loading conditions, the maximum von Mises stress in trabecular bone of the stepped screw implant model was 32 MPa and located on the notch between the first (L1 of Figure 6b) and second step sections (L2 of Figure 6b). This was approximately 17.9% lower than the stress associated with the cylindrical screw implant model (39 MPa) on the inferior notch between the first and the second threads (L1 of Figure 6a). Under vertical loading, the difference was not significant. The reason was because the absolute stress value was too small (less than 7 MPa under a vertical load of 100 N). Under horizontal loading, the difference was also not significant.

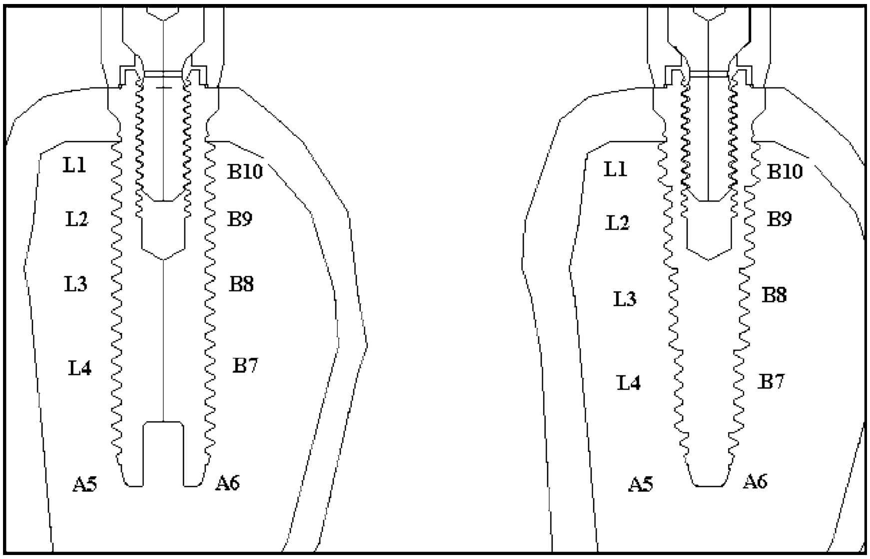

FIGURE 4. Corresponding L1 to B10 locations between the stepped screw and cylindrical screw implant along the trabecular bone-implant interface. L indicates lingual; A, apex; B, buccal.

Different cortical bone properties under an oblique load

Within the conditions of this FEA, the cortical bone quality significantly affected stress distribution. When a poor cortical bone quality was modeled using a lower elastic modulus, the stress distribution changed significantly. The location of the maximum stress adjacent to the cortical bone-implant interface moved from the superior to the inferior position (Figures 7–9).

MSC(Sup) and MSC(Inf) are the maximum von Mises stresses in the bone side of the interface between implant neck and cortical bone at the superior notch point and the inferior notch point, respectively. The maximum von Mises stress in the bone side of the interface between implant and trabecular bone is MST, which is at section L1 (cylindrical screw implant model) or the point between L1 and L2 (stepped screw implant model) indicated

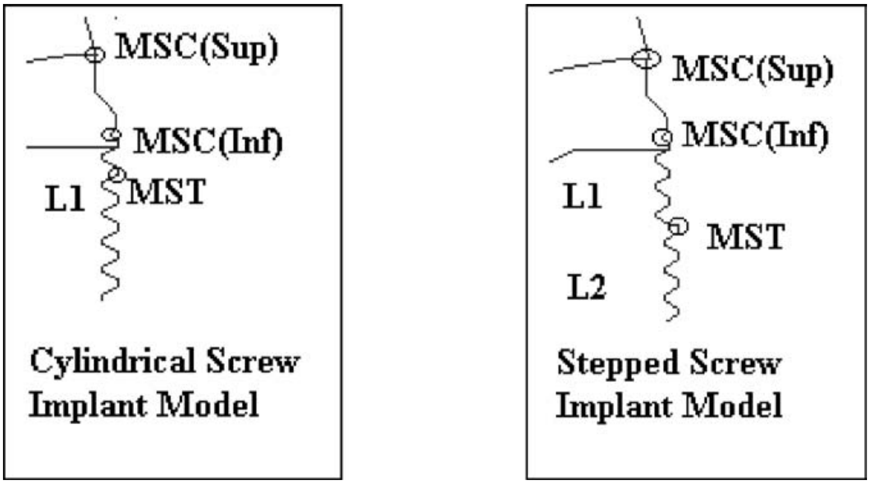

FIGURE 5. The maximum von Mises stress along the path L1 to B10 in the trabecular bone adjacent to the implant under a vertical load of 100 N, a horizontal load of 100 N, and an oblique load of 141 N at 45° inclination.

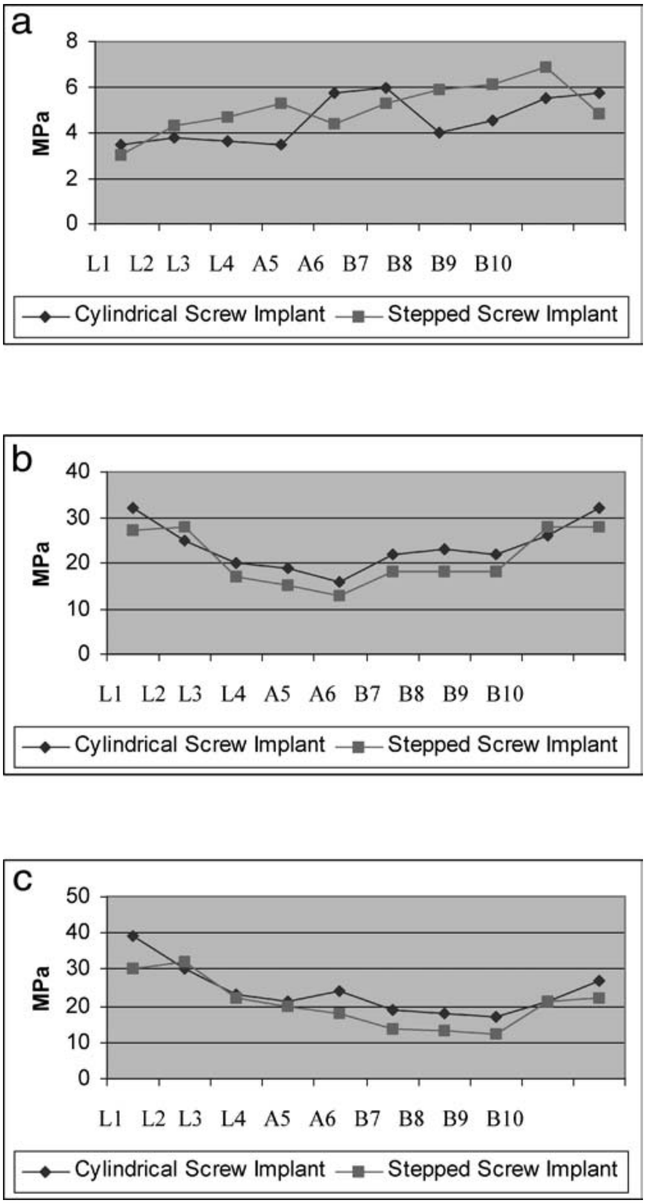

FIGURE 6. The location of the maximum stress adjacent to the bone implant interface: cylindrical screw implant (a) and the stepped screw implant (b). MSC(Sup) and MSC(Inf) indicates the maximum von Mises stress in the cortical bone adjacent to the implant neck at the superior (Sup) and inferior (Inf) notches, respectively; MST, the maximum von Mises stress in the trabecular bone adjacent to the implant, which is at the section L1 (left) or the interpoint between L1 and L2 (right).

in Figure 6. The changes in the maximum von Mises stress in cortical bone and in trabecular bone are shown in Figure 10a, b, and c. These figures reveal that MSC(Sup) decreased and MSC(Inf) and the MST increased significantly with decreasing of the cortical bone quality. Figure 10a and b shows that with relocation of the maximum stress, its level in the cortical bone was increased from 62 to 86 MPa for the stepped implant and to 90 MPa for the cylindrical implant. For the stepped implant, Figure 10c shows that the maximum stress in the trabecular bone was increased by 280%, from 32 to 90 MPa, as the elastic modulus of the cortical bone was reduced to the trabecular bone level.

For the elastic modulus of the cortical bone, it was equal to 5 GPa or higher, although the maximum stress in the cortical bone moved from a superior to an inferior notch point and the maximum stress magnitude (61 MPa) was maintained below 62 MPa for the stepped implant model. In contrast, the maximum stress magnitude in the trabecular bone was increased from 32 to 57 MPa as the elastic modulus of the cortical bone decreased from the normal level to 5 GPa. This indicates that for the stepped implant, a decrease of the cortical bone quality to a certain level (5 GPa) will not affect stress in itself, but a 78% increase in the trabecular bone will.

A comparison between the cylindrical and stepped screw implants shows that in the implant-trabecular bone interface area, the difference of maximum von Mises stress was found in models with normal cortical bone moduli (13.4 MPa). With a decrease of elastic moduli of the cortical bone to 5 GPa, the maximum stress in the trabecular bone of the stepped screw implant model was about 17.9%, 11.4%, 4.0%, and 5.0% lower than in the cylindrical implant models, respectively.

DISCUSSION

The sustainability of a dental implant is directly affected by the stress distribution and maximum stress magnitude in the bone adjacent to the implant. The stress distribution pattern gives an overview of the stress levels across the area. As there is a difference in the elastic modulus between the implant and the bone, the placement of the implant will provoke an uneven stress distribution in the area. This could change the biomechanics and physiologic environment of living bone around the implant. The maximum stress level will also affect the bone remodeling around an implant, and an overstress will obviously disintegrate the osseointegration interface between the bone and implant. To minimize these effects, an implant design optimization is, on one hand, to produce a more uniform stress pattern in the area, and on the other hand to avoid high stress in the bone. In this section, the effects of both stress distribution and maximum stress are discussed against the change of the loading conditions and bone quality.

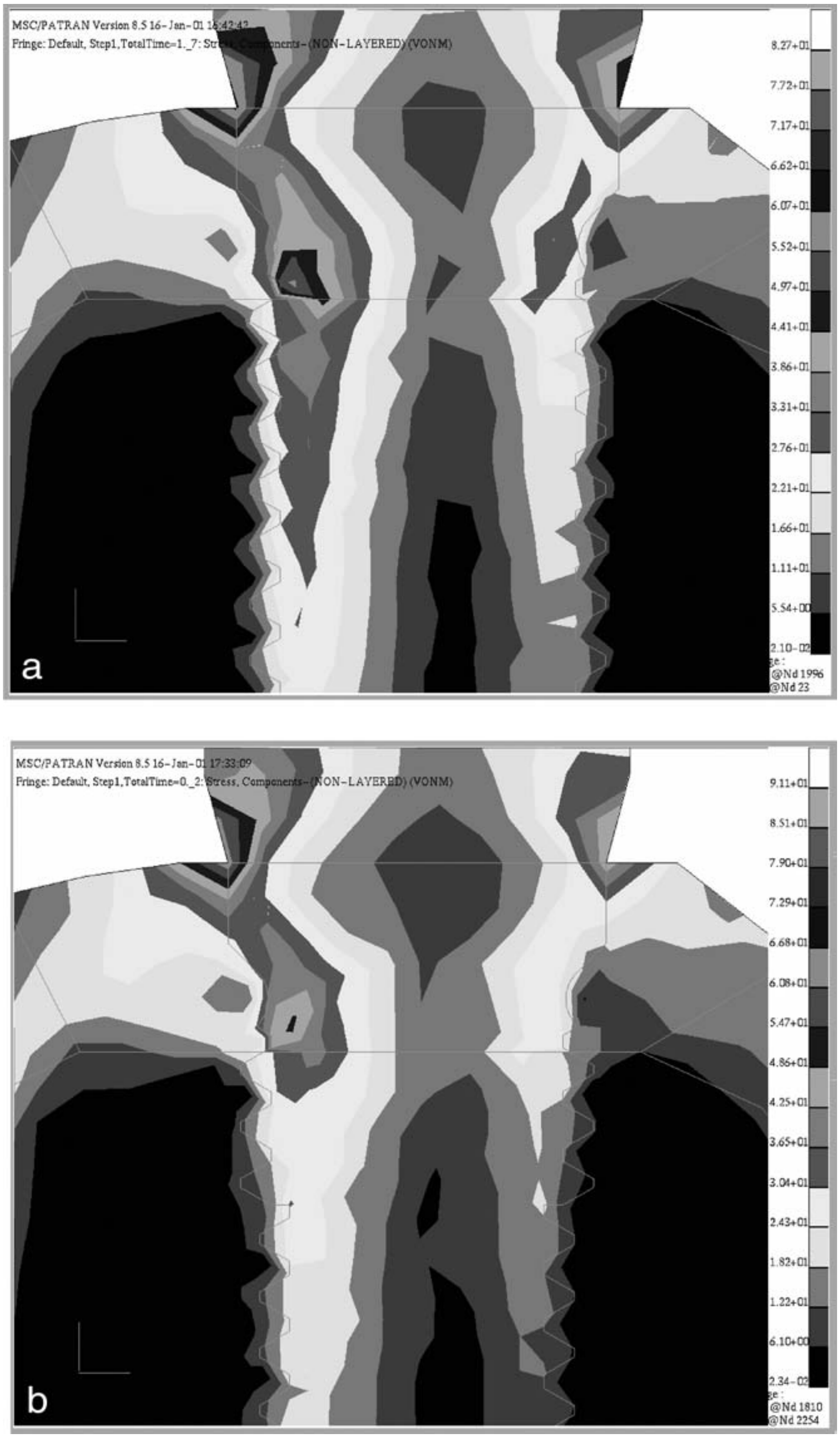

FIGURE 7 (a, b). Stress distribution in the cylindrical screw implant and in the stepped screw implant under an oblique load of 141 N in condition 1 (cortical bone E = 13.4 GPa, Poisson ratio = 0.30).

Stress distribution at the implant-bone interface

Figure 6b and c shows that under horizontal and oblique loading conditions, the stepped screw implant produces a more uniform stress distribution than the cylindrical screw implant. The maximum stress in the trabecular bone of the stepped screw implant model is 17.9% lower than that of the cylindrical screw implant model. It is believed that both an improved stress distribution and a reduced maximum stress in the trabecular bone are the result of the lower stiffness of the stepped screw implant. Holmgren et al[2] reported that a stepped press-fit implant is the most desirable design in terms of stress distribution in the surrounding bone. This study confirms that the stepped screw implant is the same.

Effects of the loading directions

The maximum von Mises stress in the bone is affected by the loading directions. Figure 6a shows that for the vertical load conditions, the absolute stress was less than 7 MPa, and the stress difference between 2 implants was relatively small. As shown in Figure 6b, a horizontal loading created a higher overall stress, but its

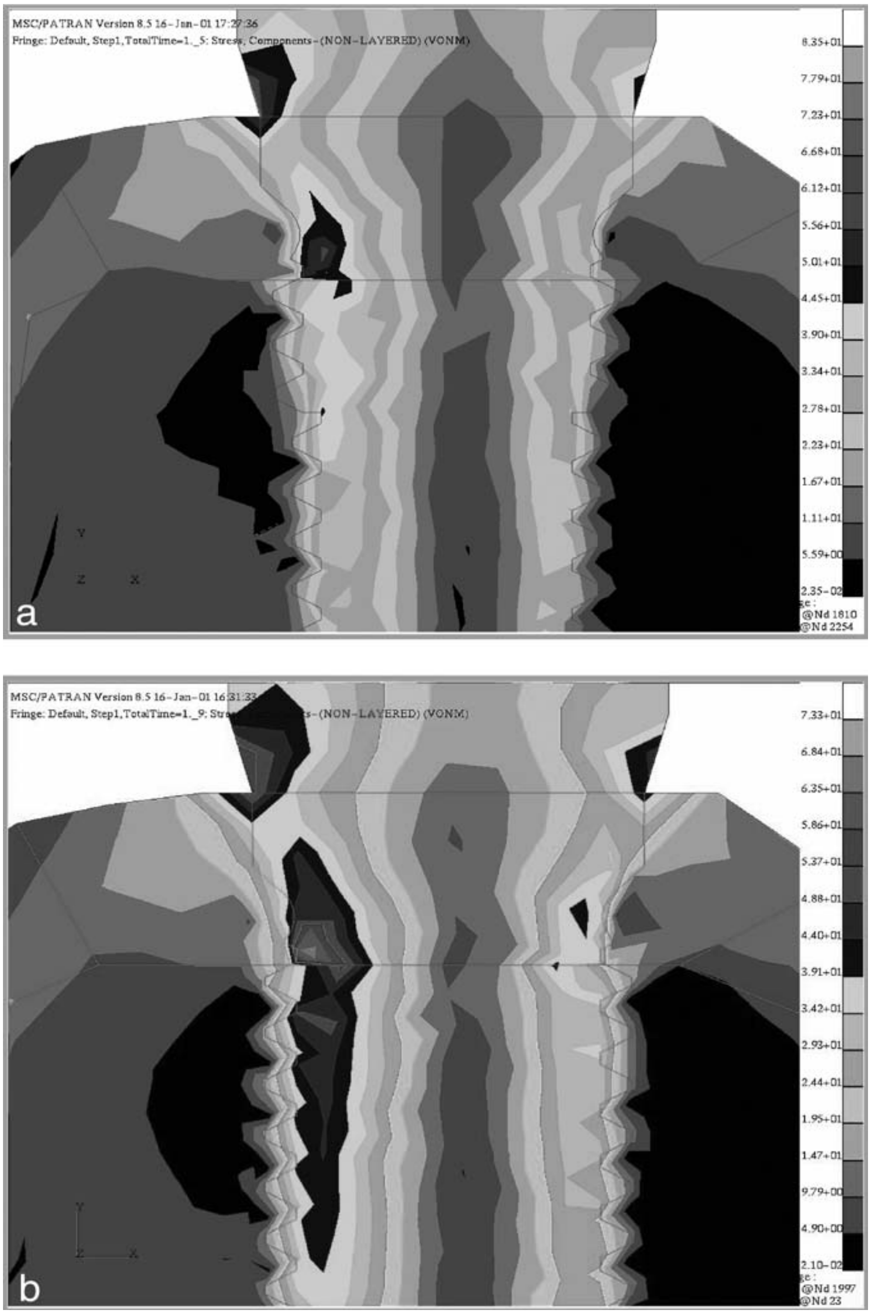

FIGURE 8. (a, b) Stress distribution in cylindrical screw implant and in stepped screw implant under oblique load of 141 N in condition 3 (cortical bone E = 7.5 GPa, Poisson ratio = 0.30).

absolute stress level was less than that produced by an oblique loading, and the maximum stress difference between the 2 models was relatively small. Figure 6c shows that an oblique load, which exists in most clinical cases, created the largest maximum stress difference in the trabecular bone. This confirms the findings of previous studies in implant loading aspects.[5,20] Based on the above evidence, the authors conclude that from the viewpoint of biomechanics, the lower maximum stress level in the stepped implant model is due to a low stiffness of the stepped implant.

Cortical bone quality

As mentioned in the introduction, FEA is a useful tool for stress simulation and has been commonly used for stress analysis of bony implant. However, in this study the FEA simulations were based on an important assumption, which was to use elastic moduli of the bone to represent the bony quality. As the other physiological properties did not have a major contribution to a linear FEA stress analysis, the following discussions will be based on such an assumption.

The result of this study shows that the varying of the cortical bone quality changes both stress distribution in the cortical bone and maximum stress in the trabecular bone. Figures 7, 8, and 9 show that when the cortical bone quality was decreased to a level similar to trabecular bone, in terms of the elastic modulus, the location of the maximum stress in the cortical bone of both cylindrical and stepped screw implant models was moved from the superior notch to the inferior notch position. In contrast, the location of the maximum stress in the trabecular bone did not change.

As shown in Figure 10a and b, the trend of repositioning of the maximum stress in the cortical bone is the same for both types of implant. With moderate decreases of the maximum stress at the superior notch, the maximum stress at the inferior notch increases sharply. Compared with the cortical bone, the trend of the maximum stress changing in the trabecular bone was similar to that at the inferior notch. However, as shown in Figure 10c, there was no significant stress difference between the 2 models when the elastic moduli of the cortical bone stayed below 10 GPa. When the elastic moduli of cortical bone was 10 GPa, the maximum stress in the stepped screw implant (39 MPa) was the same as the cylindrical screw implant in normal cortical bone with elastic moduli (13.4 GPa). Because the Branemark cylindrical screw implant had a high survival rate (98% for a 10-year period) in the normal bone condition, the results suggested that the stepped screw implant is biomechanically suitable for cortical bone with elastic moduli from 10 to 13.4 GPa. This indicates that the stepped screw implant is suitable for a poorer bone quality, which could be inadequate for a Branemark implant.

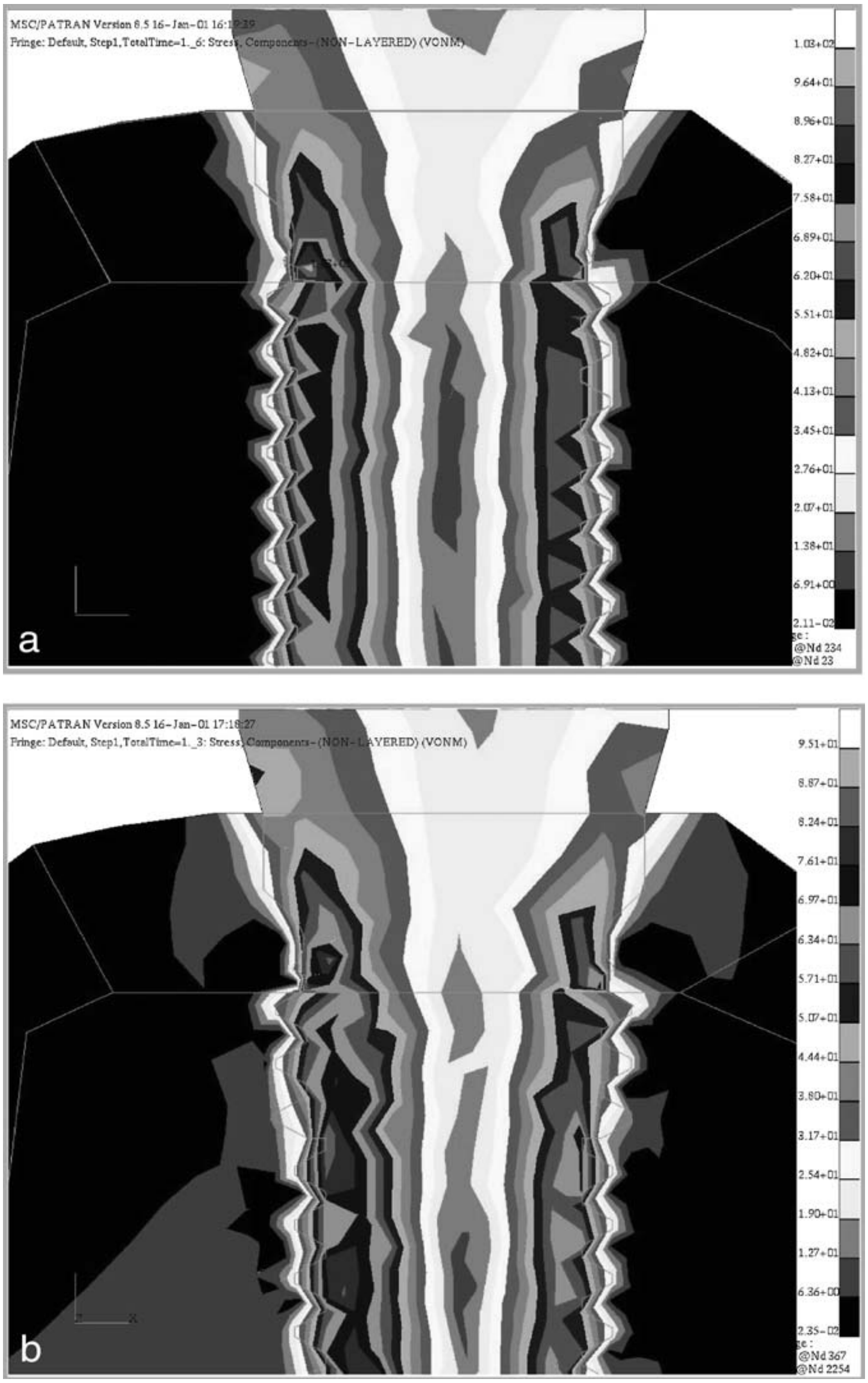

FIGURE 9. (a, b) Stress distribution in cylindrical screw implant and in stepped screw implant under oblique load of 141 N in condition 5 (cortical bone E = 1.37 GPa, Poisson ratio = 0.30; similar with trabecular bone).

CONCLUSION

This 2-dimensional FEA in vitro study indicates that compared with the cylindrical screw implant, the introduction of a stepped screw implant led to a lower maximum stress, especially for the normal cortical bone quality and oblique load conditions. The maximum stress in the trabecular bone of the stepped implant model was 17.9% less than that in the Branemark implant. It is also suggested that the stepped screw implant is biomechanically suitable for cortical bone with elastic moduli from 10 to 13.4 GPa, whereas the Branemark implant is only suitable for cortical bone with moduli above 13.4 GPa.

This study also indicates that the inclusion of an oblique load and varying of the cortical bone quality are important parameters used to simulate the biomechanical characteristics of the dental bone-implant system. Finally, the study confirms that compared with vertical and horizontal loads, the oblique load should be used for dental implant optimization studies.

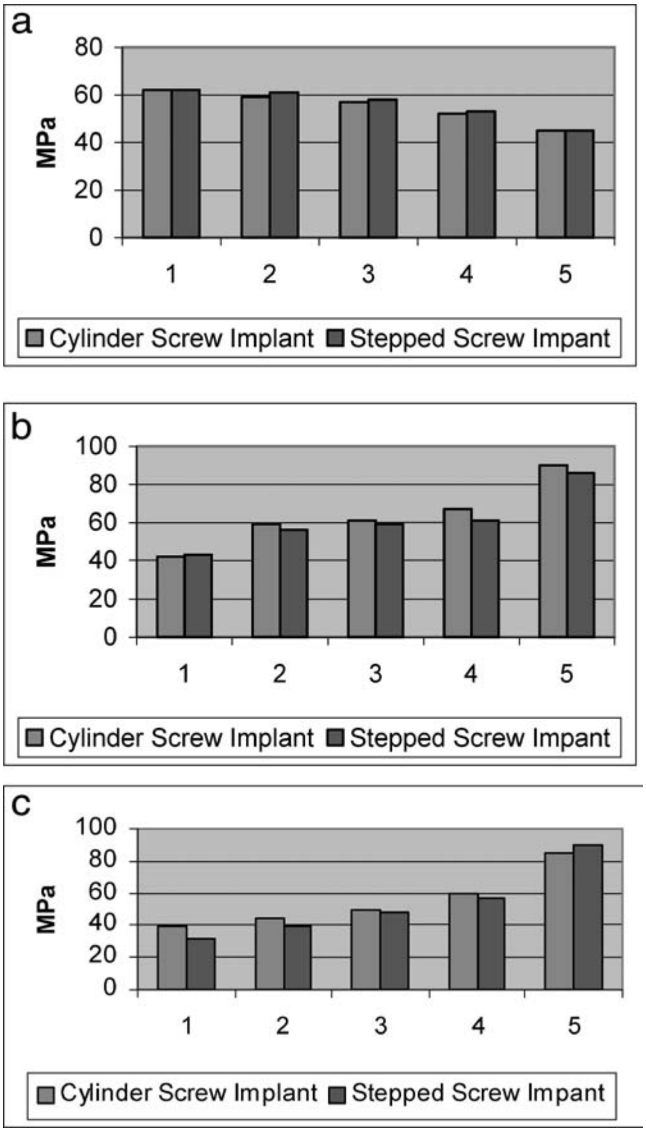

FIGURE 10. The von Mises stress of (a) MSC(sup), (b) MSC(inf), and (c) MST under an oblique load of 141 N at 45° and different cortical bone properties (conditions 1–5).

REFERENCES

1. Geng JP, Liu HC. *Exceptional Prosthodontics.* Hong Kong: Hong Kong Tranfor; 1999.

2. Holmgren EP, Seckinger RJ, Kilgren LM, Mante F. 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. *J Oral Implantol.* 1998;24:80–88.

3. Pilliar RM, Deporter DA, Watson PA, Valiquette N. Dental implant design—effect on bone remodelling. *J Biomed Mat Res.* 1991;25:467–483.

4. Vaillancount H, Pilliar RM, McCammond D. Factors affecting crestal bone loss with dental implants partially covered with a porous coating: a finite element analysis. *Int J Oral Maxillofac Implants.* 1996;11: 351–359.

5. Barbier L, Vander Sloten J, Krzesinski G, Schepers E, Van der Perre G. Finite element analysis of non-axial versus axial loading of oral implants in the mandible of the dog. *J Oral Rehabil.* 1998; 25:847–858.

6. Lum LB, Osier JF. Load transfer from endosteal implants to supporting bone: an analysis using statics. Part one: horizontal loading. *J Oral Implantol.* 1992;18: 343–348.

7. Clift SE, Fisher J, Watson CJ. Finite element stress and strain analysis of the bone surrounding a dental implant: effect of variations in bone modulus. *Proc Inst Mech Eng [H].* 1992;206: 233–241.

8. Holmes DC, Loftus JT. Influence of bone quality on stress distribution for endosseous implants. *J Oral Implantol.* 1997;23: 104–111.

9. Papavasiliou G, Kamposiora P, Bayne SC, Felton DA. Three-dimensional finite element analysis of stress-distribution around single tooth implants as a function of bony support, prosthesis type, and loading during function. *J Prosthet Dent.* 1996;76: 633–640.

10. Geng JP, Tan KBC, Liu GR. Applications of finite element analysis in implant dentistry: a review of literatures. *J Prosthet Dent.* 2001;85:585–598.

11. Van Oosterwyck H, Duyck J, Vander S, et al. The influence of bone mechanical properties and implant fixation upon bone loading around oral implants. *Clin Oral Implant Res.* 1998;9:407–418.

12. Mailath G, Stoiber B, Watzek G, Matejka M. Bone resorption at the entry of osseointegrated implants—a biomechanical phenomenon. Finite element study. *Z Stomatol.* 1989; 86:207–216.

13. Matsushita Y, Kitoh M, Mizuta K, Ikeda H, Suetsugu T. Two-dimensional FEA analysis of hydroxyapatite implants: di-

ameter effects on stress distribution. *J Oral Implantol.* 1990;16: 6–11.

14. Stellingsma C, Meijer HJ, Raghoebar GM. Use of short endosseous implants and an overdenture in the extremely resorbed mandible: a five-year retrospective study. *J Oral Maxillofac Surg.* 2000;58:382–388.

15. Siegele D, Soltesz U. Numerical investigations of the influence of implant shape on stress distribution in the jaw bone. *Int J Oral Maxillofac Implants.* 1989;4:333–340.

16. Cowin SC. *Bone Mechanics.* Boca Raton, Fla: CRC Press; 1989.

17. Cook SD, Klawitter JJ, Weinstein AM. A model for the implant-bone interface characteristics of porous dental implants. *J Dent Res.* 1982;61:1006–1009.

18. Borchers L, Reichart P. Three-dimensional stress distribution around a dental implant at different stages of interface development. *J Dent Res.* 1983;62: 155–159.

19. Xu W, Crocombe AD, Hughes SC. Finite element analysis of bone stress and strain around a distal osseointegrated implant for prosthetic limb attachment. *Proc Inst Mech Eng [H].* 2000;214: 595–602.

20. Rieger MR, Fareed K, Adams WK, Tanquist RA. Bone stress distribution for three endosseous implants. *J Prosthet Dent.* 1989;61:223–228.

21. Ronald LS, Svenn EB. Nonlinear contact analysis of preload in dental implant screws. *Int J Oral Maxillofac Implants.* 1995; 10:295–302.