Effect of Dental Implant Cross-Sectional Design on Cortical Bone Structure Using Finite Element Analysis

Osama Abu-Hammad, BDS, MSc, PhD;* Ameen Khraisat, BDS, PhD;†
Najla Dar-Odeh, BDS, FDS RCS (Ed);‡ Mohammed El-Maaytah, BDS, MSc, PhD, FDS RCPS (Glasg), LDS RCS (Eng)§

ABSTRACT

Purpose: This finite element analysis investigation evaluated the effect of different implant cross-sectional designs on bone stress levels under different loading patterns.

Materials and Methods: Finite element analysis program was used to construct four different three-dimensional models describing 4 × 10-mm implants in blocks of cortical and trabecular bone. A 5-mm-long abutment was modeled above each implant. The implant in model 1 was unthreaded, while in model 2 the implant was circularly threaded. The third implant in model 3 had the cross-sectional shape as a 16-sided star-shaped design. The implant in model 4 was constructed unthreaded, with a diameter of 4.5 mm. Vertical and horizontal loads of 100 N each were applied on the top middle node of each implant assembly. All nodes at the bottom surface of the bone models were restrained.

Results: By comparing models 1, 2, and 3, the lowest bone stress values under vertical and horizontal forces were observed around the unthreaded implant in model 1 (8.92 and 94.52 MPa, respectively). The highest stress value under vertical loading was shown around the threaded implant in model 2 (10.07 MPa), whereas the highest stress value under horizontal loading was observed around the star-shaped implant in model 3 (108.40 MPa). Model 4, with a wider unthreaded design, had stress values under vertical and horizontal loading of 7.32 and 71.35 MPa, respectively.

Conclusions: It was concluded that the unthreaded implant design produced the least bone stress. An increase in implant diameter could produce marked reduction in stress value in the bone around the neck of the implant.

KEY WORDS: implant design, implant thread, bone stress, strain, implant diameter, finite element analysis

INTRODUCTION

Stress build-up in the bone around the dental implant is believed to have a major role in the phenomenon of bone resorption around the necks of dental implants.[1–3] One of the suggested mechanisms is what happens when the elastic limit of the bone around the implant is surpassed on the implant, initiating microfractures in the bone and, thus, bone resorption.[3]

The implant design, diameter, material properties, and surface criteria are among the factors that influence the stress in the surrounding bone.[4–6] Other factors such as bone volume and shape and the material properties of the bone can influence the amount of bone stresses.[7,8] However, the primary stability of the implant is another important factor for osseointegration to take place. It is achieved by taper locking the implant in the prepared socket or through the engagement of implant threads to prepared taps in the socket.[9–11]

However, the current implant market is nourished by a variety of implant designs that are periodically being modified or even completely changed.[8–12]

*Associate professor, Department of Conservative Dentistry and Prosthodontics, Faculty of Dentistry, University of Jordan, Amman, Jordan; †assistant professor, Department of Conservative Dentistry and Prosthodontics, Faculty of Dentistry, University of Jordan, Amman, Jordan; ‡associate professor, Department of Oral Medicine, Oral Surgery and Periodontics, Faculty of Dentistry, University of Jordan, Amman, Jordan; §Clinical lecturer, Oral and Maxillofacial Surgery/Head and Neck Unit, University College London Hospitals, London; Division of Maxillofacial, Diagnostic, Medical and Surgical Sciences, Eastman Dental Institute, London, UK

Reprint requests: Dr. Ameen Khraisat, Salt P.O.B. 436, 19110 Jordan; e-mail: khraisat@lycos.com

DOI 10.1111/j.1708-8208.2007.00048.x

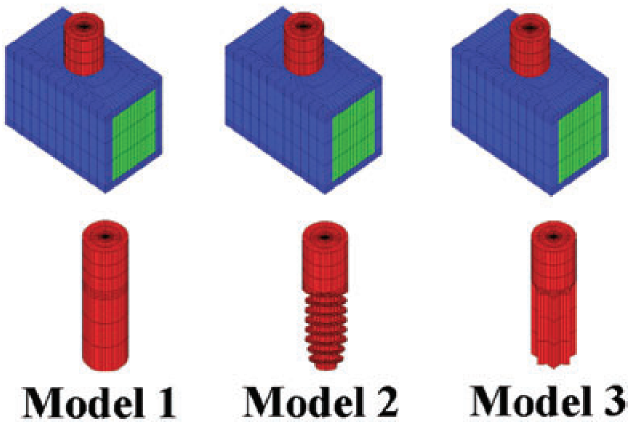

Figure 1 Models 1 to 3 along with the individual implant below.

The aim of this finite element analysis study was to evaluate and compare the effects of different implant cross-sectional designs on the surrounding bone stress and strain levels under different loading patterns. Furthermore, the effect of implant diameter change on the surrounding bone stress and strain values was investigated.

MATERIALS AND METHODS

The finite element analysis program NISA (Cranes Software Inc., Troy, MI, USA), was used in this investigation to construct different three-dimensional models.

Three models were constructed describing implants with 4-mm external diameters and 10-mm lengths (Figure 1). These modeled implants were installed in identical quadrilateral pieces of modeled bone of 14-mm lengths, 8-mm widths, and 11-mm heights. The pieces of bone were composed of trabecular bone covered with a 1-mm-thick layer of cortical bone. The cortical bone layer covered all aspects of the trabecular bone corresponding to the upper and lower borders of the body of the mandible and its buccal and lingual surfaces, as illustrated in Figure 1. A 5-mm-long abutment with the same diameter as the implant was modeled attached above each implant.

The implant in model 1 was designed as an unthreaded cylinder. In model 2, the implant was threaded with circular threads (rather than helical) in a way to simplify the model construction. The threads were modeled as radial disk-shaped extensions (0.2 mm) of the main body of the implant, as illustrated in Figure 1. The third implant in model 3 had a cross-sectional shape as a 16-sided star-shaped design (eight longitudinal side angles) that would theoretically intensify the stress and strain values in the surrounding bone structure (see Figure 1). A fourth model was constructed with unthreaded implant similar to that of model 1, but with a diameter of 4.5 rather than 4 mm.

All materials representing the different regions of each model were assumed homogenous, isotropic, and linearly elastic. Material properties used in the study modeling were similar to those used in a previous study.[13] All models were constructed using three-dimensional 20-noded hexahedral brick elements. All nodes were merged to assume perfect bond between all regions in the model including the interface between the implant and the bone.

Loads on each implant were modeled as separate vertical and horizontal point loads of 100 N each applied on the top middle node of each implant/abutment assembly in the models. The vertical force was directed downward along the long axis of the implant, whereas the horizontal force was applied toward the side of the model describing its buccal or lingual aspect. All nodes at the bottom surface of the bone models were restrained. Compressive stresses and resultant strains were recorded only in the models for each loading case in the crestal bone around the necks of the implants. This region is thought to display the majority of damaging stresses in the bone around the implant.[1–5]

RESULTS

Stress and strain values under vertical and horizontal loading in the bone around the neck region of implants in all models were registered (Table 1).

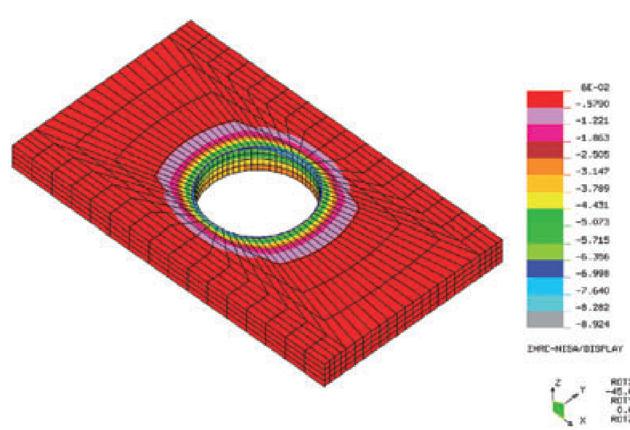

Figure 2 Compressive stresses in the crestal bone around the neck of the dental implant in model 1 under vertical loading. Stresses are displayed in different colors in the original model. Each color represents a given stress value as indicated in the color column on the right side.

TABLE 1 Maximum Stress and Strain Values in the Bone around the Necks of Dental Implants in the Four Models under Vertical and Horizontal Loading

	Vertical loading		Horizontal loading	
Models	Compressive stresses (MPa)	Resultant strains ($\times 10^{-4}$ mm/mm)	Compressive stresses (MPa)	Resultant strains ($\times 10^{-4}$ mm/mm)
1	8.92	9.11	94.52	241.30
2	10.07	17.20	101.00	315.00
3	9.82	11.50	108.40	274.50
4	7.32	7.42	71.35	203.60

When comparing models 1, 2, and 3, the results of compressive stress values under vertical loading ranged from 8.92 (in model 1) to 10.07 MPa (in model 2) (Figures 2–6). The results of compressive stress values under horizontal loading ranged from 94.52 (in model 1) to 108.40 MPa (in model 3) (Figure 3, 5, and 7). The resultant strains under vertical loading ranged from 9.11×10^{-4} (in model 1) to 17.2×10^{-4} mm/mm (in model 2). Under horizontal loading, the resultant strain results ranged from 241.30×10^{-4} (in model 1) to 315.00×10^{-4} mm/mm (in model 2). Therefore, the lowest bone stress and strain values under vertical and horizontal forces were observed around the unthreaded cylindrical implant in model 1. The highest stress value under vertical loading was shown around the threaded implant in

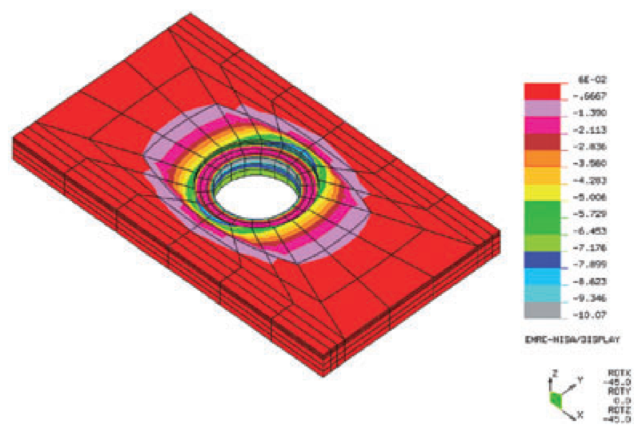

Figure 4 Compressive stresses in the crestal bone around the neck of the dental implant in model 2 under vertical loading. Stresses are displayed in different colors in the original model. Each color represents a given stress value as indicated in the color column on the right side.

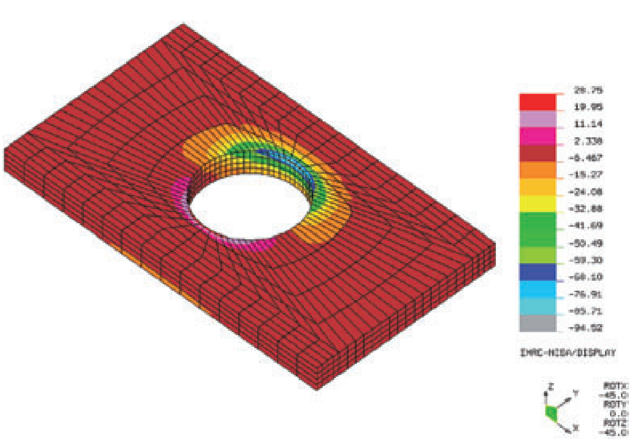

Figure 3 Compressive stresses in the crestal bone around the neck of the dental implant in model 1 under horizontal loading. Stresses are displayed in different colors in the original model. Each color represents a given stress value as indicated in the color column on the right side.

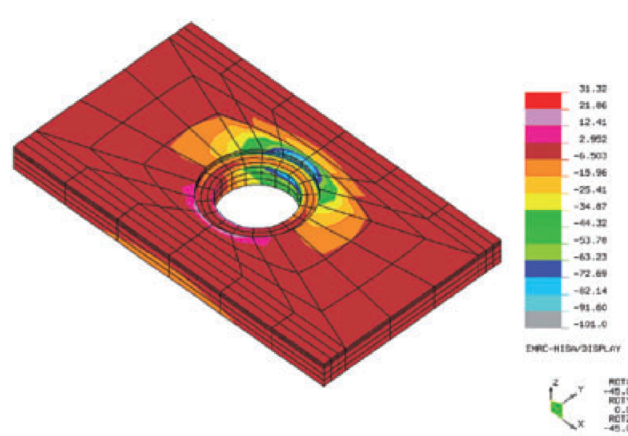

Figure 5 Compressive stresses in the crestal bone around the neck of the dental implant in model 2 under horizontal loading. Stresses are displayed in different colors in the original model. Each color represents a given stress value as indicated in the color column on the right side.

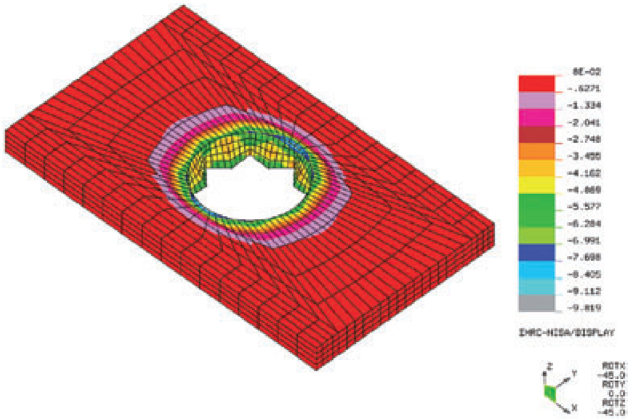

Figure 6 Compressive stresses in the crestal bone around the neck of the dental implant in model 3 under vertical loading. Stresses are displayed in different colors in the original model. Each color represents a given stress value as indicated in the color column on the right side.

model 2, whereas the highest stress value under horizontal loading was observed around the star-shaped implant in model 3. The highest strain values under vertical and horizontal loading were observed in the bone around the threaded implant in model 2 (see Table 1).

Regarding model 4 with a wider unthreaded cylindrical design, compressive stress values under vertical and horizontal loading were 7.32 and 71.35 MPa, respectively. The resultant strain values under vertical and horizontal loading were 7.42×10^{-4} and 203.60×10^{-4} mm/mm, respectively (see Table 1).

DISCUSSION

The result of this study showed that compressive stress and strain values under horizontal loading were about 10 and 20 times higher than those under vertical loading, respectively.

For the first three models, the unthreaded cylindrical implant generated the lowest bone compressive stress and strain values under vertical and horizontal loading. Generally, the threaded implant generated the highest stress and strain values in the bone. Probably, that can be attributed to its threads that are extending laterally inside the bone and, thus, would exert compression on the bone under, specifically, vertical loading. Under horizontal loading, the threads would form the frontline of stress concentration, and that might have been the reason for the highest strain value that resulted from the tested models. The star-shaped implant generated the highest compressive stresses under horizontal loading. This can be attributed to the vertical orientation of the sharp line angles located at its sides. These should exert the most damaging effect on the bone upon application of bending forces on the implant.[14,15]

Under vertical loading, the unthreaded implant with a wider diameter (model 4) showed lower compressive stress and strain values by about 18% than those with a smaller diameter (model 1). On the other hand, the compressive stress and strain values under horizontal loading were 24 and 16% less with a wider implant diameter than their corresponding values with the narrower implant, respectively. These results indicate the importance of increasing the diameter in reducing bone stress and strain values. The unthreaded cylindrical implant might be considered a neutral design with regard to shape and design, and it had been employed in similar studies.[16–19]

The present study did not aim to predict the behavior of bone in reality; it rather explored the behavior of the given implant designs inside standardized pieces of bone. The shape of the piece of bone was assumed as quadrilateral for simplification in this study. Moreover, the modeling of the implant/bone interface was performed with the assumption of a perfect bond. This was consistent with that of other studies on implant-removal torques and the pattern of bone behavior around the implant.[18,20]

The values chosen in this study for the material properties of the different regions of the models were previously reported and used in similar finite element studies.[13,16,20] In fact, the huge variations in the reported values of properties in dental literature emphasize the need for consensus on this subject.[21,22]

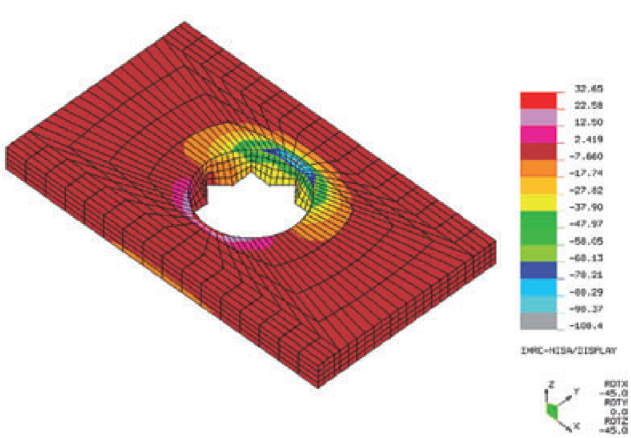

Figure 7 Compressive stresses in the crestal bone around the neck of the dental implant in model 3 under horizontal loading. Stresses are displayed in different colors in the original model. Each color represents a given stress value as indicated in the color column on the right side.

Bone safety that was brought by the usage of unthreaded cylindrical implant design might signify the benefit of such design in several fields of dentistry. The relatively low bone stress values under horizontal loading would highlight the application of this design as an orthodontic anchorage, where tension is regularly present.

CONCLUSIONS

Within the limitations of this study, it can be concluded that the unthreaded implant design produced the least bone stress and strain under vertical and horizontal loading when compared with the threaded implant. An increase in implant diameter could produce marked reduction in stress and strain values in the bone around the neck of the implant.

REFERENCES

1. Vaillancourt H, Pilliar RM, McCammond D. Finite element analysis of crestal bone loss around porous-coated dental implants. J Appl Biomater 1995; 6:267–282.
2. Lai H, Zhang F, Zhang B, Yang C, Xue M. Influence of percentage of osseointegration on stress distribution around dental implants. Chin J Dent Res 1998; 1:7–11.
3. 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.
4. Hamdan M, Blanco L, Khraisat A, Tresguerres IF. Influence of titanium surface charge on fibroblast adhesion. Clin Implant Dent Relat Res 2006; 8:32–38.
5. Heckmann SM, Linke JJ, Graef F, Foitzik Ch, Wichmann MG, Weber HP. Stress and inflammation as a detrimental combination for peri-implant bone loss. J Dent Res 2006; 85:711–716.
6. Khraisat A, Abu-Hammad O, Al-Kayed AM, Dar-Odeh N. Stability of the implant/abutment joint in a single-tooth external-hexagon implant system: clinical and mechanical review. Clin Implant Dent Relat Res 2004; 6:222–229.
7. Sevimay M, Turhan F, Kilicarslan MA, Eskitascioglu G. Three-dimensional finite element analysis of the effect of different bone quality on stress distribution in an implant-supported crown. J Prosthet Dent 2005; 93:227–234.
8. Taylor TD, Agar JR, Vogiatzi T. Implant prosthetics: current perspective and future directions. Int J Oral Maxillofac Implants 2000; 15:66–75.
9. Gentile MA, Chuang SK, Dodson TB. Survival estimates and risk factors for failure with 6 × 5.7-mm implants. Int J Oral Maxillofac Implants 2005; 20:930–937.
10. Sakoh J, Wahlmann U, Stender E, Nat R, Al-Nawas B, Wagner W. Primary stability of a conical implant and a hybrid, cylindric screw-type implant in vitro. Int J Oral Maxillofac Implants 2006; 21:560–566.
11. Motoyoshi M, Yano S, Tsuruoka T, Shimizu N. Biomechanical effect of abutment on stability of orthodontic mini-implant. A finite element analysis. Clin Oral Implants Res 2005; 16:480–485.
12. Quek CE, Tan KB, Nicholls JI. Load fatigue performance of a single-tooth implant abutment system: effect of diameter. Int J Oral Maxillofac Implants 2006; 21:929–936.
13. Abu-Hammad OA, Harrison A, Williams D. The effect of a hydroxyapatite-reinforced polyethylene stress distributor in a dental implant on compressive stress levels in surrounding bone. Int J Oral Maxillofac Implants 2000; 15:559–564.
14. Rangert B, Jemt T, Jörnéus L. Forces and moment on Brånemark implants. Int J Oral Maxillofac Implant 1989; 4: 241–247.
15. Rangert B, Sullivan RM, Jemt TM. Load factor control for implants in the posterior partially edentulous segment. Int J Oral Maxillofac Implants 1997; 12:360–370.
16. Skalak R, Zhao Y. Similarity of stress distribution in bone for various implant surface roughness heights of similar form. Clin Implant Dent Relat Res 2000; 2:225–230.
17. Park SH, Wang HL. Implant reversible complications: classification and treatments. Implant Dent 2005; 14:211–220.
18. Natali AN, Meroi EA, Williams KR, Calabrese L. Investigation of the integration process of dental implants by means of a numerical analysis of dynamic response. Dent Mater 1997; 13:325–332.
19. Abu-Hammad O, Khraisat A, Dar-Odeh N, Jagger DC, Hammerle CHF. The staggered installation of dental implants and effect on bone stresses. Clin Implant Dent Relat Res (In revision).
20. Carlsson L, Rostlund T, Albrektsson B, Albrektsson T. Removal torques for polished and rough titanium implants. Int J Oral Maxillofac Implants 1988; 3:21–24.
21. Geng JP, Tan KB, Liu GR. Application of finite element analysis in implant dentistry. A review of the literature. J Prosthet Dent 2001; 85:585–598.
22. Takuma M, Tsutsumi S, Kurokawa F, Takashima F, Miyauchi S, Maruyama T. The influence of materials difference on stress distribution and bone remodeling around alumina and titanium dental implants. J Osaka Univ Dent Sch 1990; 30:86–96.