Model-based biomechanical dental implant optimization in bone-implant system

Jian-Ping Geng, BDS, MSD, PhD,[a] Wei Xu, PhD,[b] and Wei-Qi Yan, MD, PhD[a]

[a]Clinical Research Center, The Second Affiliated Hospital, Zhejiang University School of Medicine, Hangzhou, P. R. China
[b]Centre for Biomedical Engineering, School of Engineering, University of Surrey, Guildford, UK

Purpose: The purpose of this study was to identify the important parameters for a model-based design optimization method through comparison of different biomechanical environments and various dental implant models.
Materials and Methods: Finite element analysis (FEA) was applied to model a mandibular bone implant system. Essential parameters, such as, cortical bone properties, boundary conditions, and loading directions, were investigated and altered. The effects of altering these parameters in bone-implant FEA models were evaluated based on differences in stresses.
Results: The results of this study showed that under different oblique loads, cortical bone properties and constraints the differences in the stress level and distribution between the different implant designs could be clearly identified. Key parameters for a model-based design optimization method were identified effectively by using FEA to compare the stress situation in different implant systems with various biomedical environments.
Conclusion: Without knowing which parameters play substantial roles, design optimization of a dental implant system remains a challenge, especially when comparison of different implants are involved and bone quality has to be taken into account. This study shows that an oblique load, excellent cortical bone property, and minimal support constraints are critical parameters affecting the stress distribution in the bone-implant model. Therefore these parameters were important for implant design optimization. **(Int Chin J Dent 2007; 7: 15-22.)**
Key Words: bone, finite element analysis, implant.

Introduction

It has been nearly 40 years since the first dental implant was placed in an edentulous patient.[1] Dental implants are effectively used to support prostheses for a variety of tooth loss scenarios. Although the traditional undisturbed healing concept has been clinically proven to be successful in long-term for both single[2,3] and 2-stage[4,5] treatment protocols, recent promising results have also been observed when implants were subjected to immediate functional loads.[1,6] These clinical findings support a scenario that the osseointegration and prognosis after the placement of an implant depend on neither traditional undisturbed healing concepts nor recent practice of immediate application of functional load after implantation. A successful osseointegration implant is subject to an appropriate biomechanical environment, which effects the material properties of the implant and prosthesis, implant geometry, surface structure, quality and quantity of the surrounding bone, and nature of the bone-implant interface.[7] A report has indicated that the biomechanical environment has a strong effect to the long-term maintenance of the interface between implant and bone.[8]

Currently, there is a wide range dental implant systems available on the market. Although the success rates of some implant systems have been high, implant failures do occur.[9,10] Since the initial survival of the dental implant requires a healthy stress level at the bone-implant interface, certain parameters, such as load direction and bone quality have substantial effects to the performance of a dental implant.[11,12] Better understanding of these effects will provide a basis for optimizing an implant design with improved biomechanical stability and further enhance the clinical success.

Many assumptions have been investigated, including (a) the geometric detail of the object to be modeled, (b) the material properties, and (c) the applied boundary conditions. According to the previous reports,[7,13-17] to include these assumptions in a biomechanically optimized dental implant was a complicated task. Inappropriate application of these assumptions to the FE model was one of the reasons that different models produced different optimal implant designs. Judging which implant design is better in terms of biomechanics remains a challenge. In order to solve the problem, the authors carried out model based parameter studies to identify the crucial factors for dental implant optimization. The study is based on various parameters such as the implant type and bone conditions. No considerations were given to whether specific implant parameters were suitable for a particular bone quality. This built a complete picture for better understanding the effects of the parameters. The results revealed crucial evidence for model based analyses of dental implant in terms of biomechanics. The studies were conducted using a standard mandibular model with different implant parameters, load directions, cortical bone quality and boundary conditions.

Materials and Methods

A 2-dimensional (2D) finite element model of an implant-bone system was developed using FEA software (MSC/PATRAN 8.5; MSC Software Corp., Santa Ana, CA, USA) as shown in Fig. 1. It shows three different bone-implant systems and boundary conditions (A), cylindrical screw implant model, base-support type, (B), V-type stepped screw implant model, base-support type, (C), V-type stepped screw implant model, middle-support type.

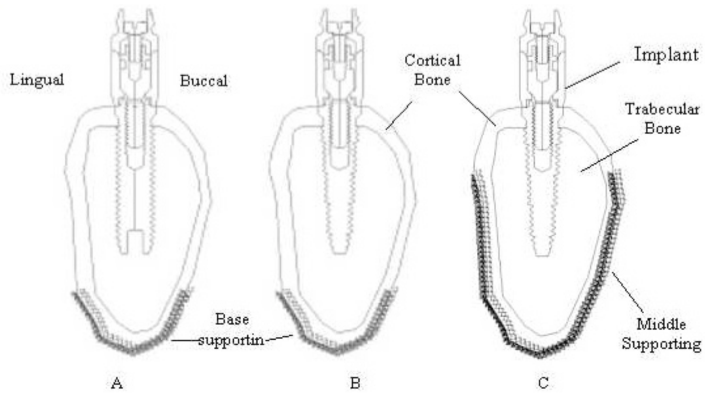


Fig. 1. Different bone-implant systems and boundary conditions.

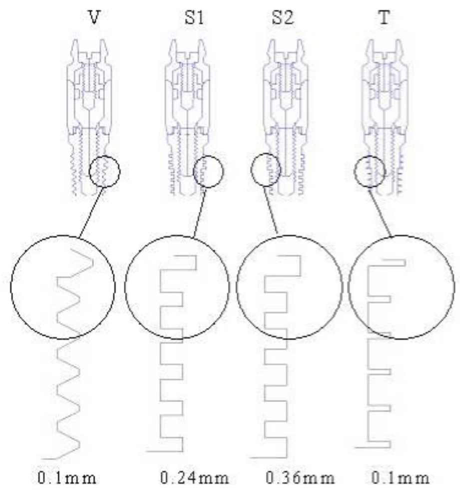


Fig. 2. Four different stepped screw implants (left) and thread configurations (right).

Five implants were modeled, including one cylindrical screw implant (Fig. 1A, Branemark Implant; Nobel Biocare AB, Goteborg, Sweden) and four single stepped screw experimental implants with different thread profiles, as shown in Fig. 2 (GJP Implants, National Biomaterials Centre, Chengdu, P. R. China). Both the cylindrical and stepped screw implants had equivalent length (15 mm), neck diameter (4.1 mm), and similar numbers of threads, step position, and thread pitch. As shown in Fig. 2, the stepped screw implants had four different thread profiles: truncated v-thread (V), thin-thread (T), and two square-thread profiles with different thread widths, 0.24 mm (S1) and 0.36 mm (S2). Cortical and cancellous bone were modeled from a patient's CT image,[18] representing the standard buccal-lingual cross-section of the posterior human mandible. The thickness of the cortical bone around the implant neck was 2 mm.

Compared with the 3D counterpart, the use of a 2D finite element model was based on the fact that a proper 2D model was more efficient in terms of model generation and use of computational resource. The results could be considered as accurate if a qualitative study was involved.[19] Plane strain analysis was used for structures in which one dimension was larger than the other two dimensions, and the cross-section of interest was perpendicular to the long axis. It was used by the authors for this study because it was regarded as the best simplification for a human mandibular model since the modeled cross-section represented the principal loading direction of the teeth. In this analysis, therefore, a plane strain assumption was adopted and a representative buccal-lingual cross-section of the posterior human mandible was modeled. The implant thickness was defined as 4 mm in this study to avoid potential inaccuracy due to buccal-lingual horizontal and oblique load.[18] The finite element model was meshed using Quad 4 (four nodes) elements with edge length of 0.5 mm (defined using the Global Edge Length by the FEA software). At the interface between the implant and bone, the global edge length of the element was refined to 0.1 mm. To determine suitable global edge lengths a pilot study was carried out. As the von Mises stress is commonly used to characterize FEA studies 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 cylindrical implant models. A nodal force was applied on the top of the transmucosal abutment. The nodes over the free edges of cortical bone were constrained in the x, y, and z directions rigidly without movement. The three boundary conditions of the implant-bone systems are shown in Fig. 1. Young's Modulus (MPa) and Poisson's ratio are 117,000 and 0.30 for the CPT implant, 13,400 and 0.30 for cortical bone,[20] and 1,370 and 0.31 for trabecular bone.[21] In terms of implant-bone contact, it was assumed that the osseointegration between bone and implant was 100%.

In this study, three parameters were investigated and altered in the standard mandibular bone-implant finite element model to determine the parameters that had a significant effect to the stress conditions among five different models. They were load direction, cortical bone quality, and boundary conditions. Different load directions in normal cortical bone condition were compared between a cylindrical screw and the single stepped screw implants. Three loading situations were considered: (1) vertical load of 100 N, (2) horizontal load of 100 N, and (3) combined oblique load of 141 N at 45-degree inclination. Use of the inclination load was based on a previous comparative study of the effects of implant shape and load direction to the osseointegrated dental implants.[10]

Different cortical bone properties under the oblique load condition were compared between a cylindrical screw implant and a single stepped screw implant. Clinically, the Young's modulus of cortical bone varies from 13,400 MPa (a normal bone) to 1,370 MPa (a weak bone) which is similar to the trabecular bone. In this comparative study, five different conditions (Young's modulus in MPa and Poisson's ratio) were modeled; condition 1 (13,400, 0.30); condition 2 (10,000, 0.30), condition 3 (7,500, 0.30), condition 4 (5,000, 0.30), and condition 5 (1,370, 0.31). Different boundary conditions under the oblique load and normal cortical bone condition were compared among the four single stepped screw implants. In this study, different boundary conditions were assigned to the models by changing constraints that support the mandible from the middle (Fig. 1C) to base (Fig. 1B) bone segment.

Results

The von Mises stresses from the FEA are shown in Figs. 3 and 4. Comparing the V-type stepped screw implant with the cylindrical screw implant models, the result demonstrated that in the cortical bone immediately adjacent to the implant neck, there were no differences in stress distribution. Under 100 N vertical, 100 N horizontal, and 141 N oblique (45° inclination) three different loading conditions, the maximum stress in the cortical bone of two models was approximately at the same level (8 MPa, 59 MPa, and 61 MPa respectively).

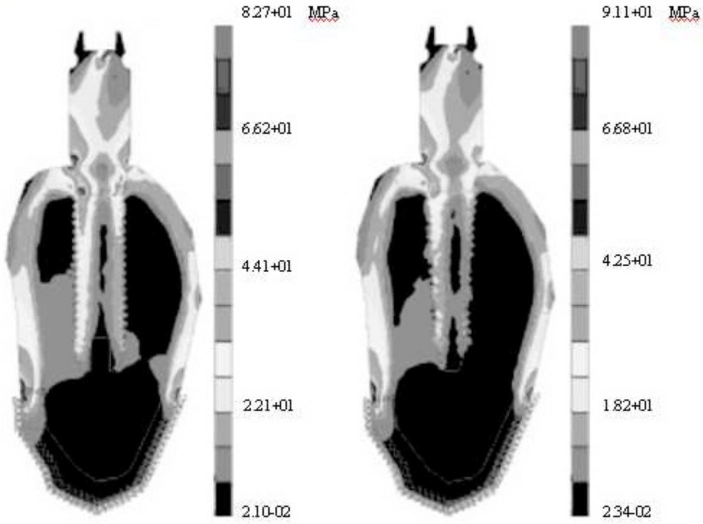


Fig. 3.
Stress distributions under an oblique load of 141 N with normal cortical bone (left) cylindrical screw implant, and (right) stepped screw implant.

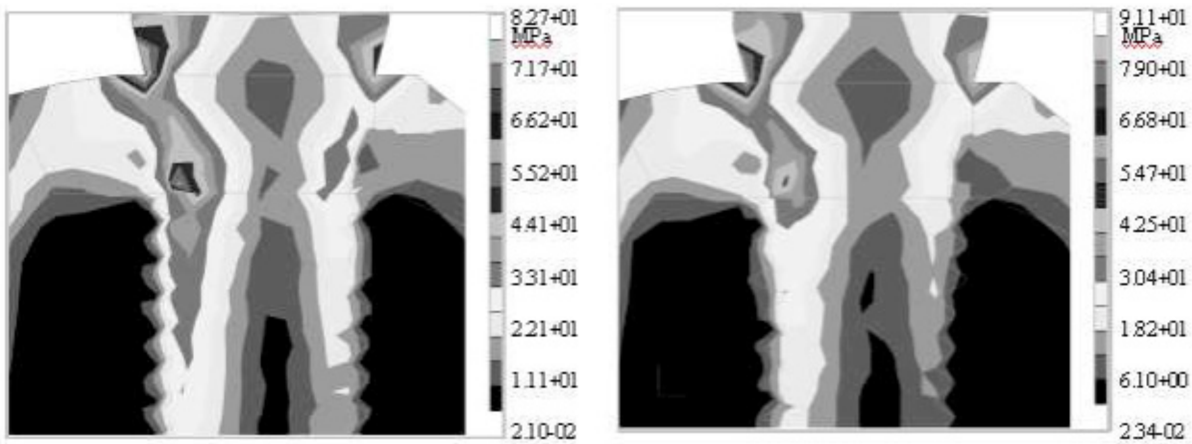


Fig. 4. Stress distribution of cylindrical screw implant (left) and of stepped screw implant (right) under oblique load of 141 N with condition 1 (cortical bone of E=13,400 MPa and P=0.30).

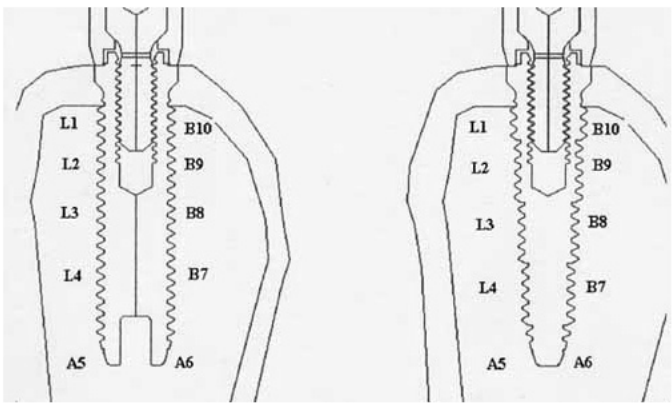
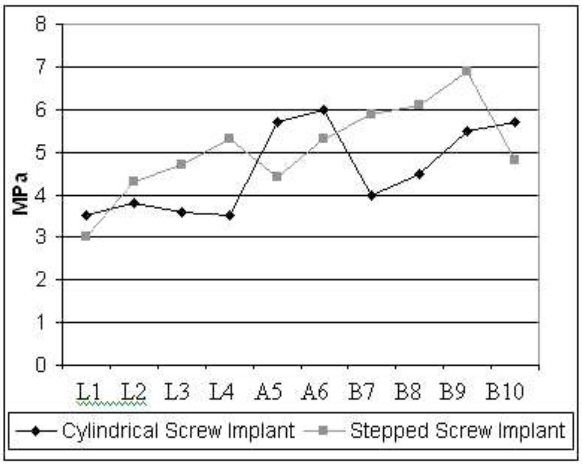
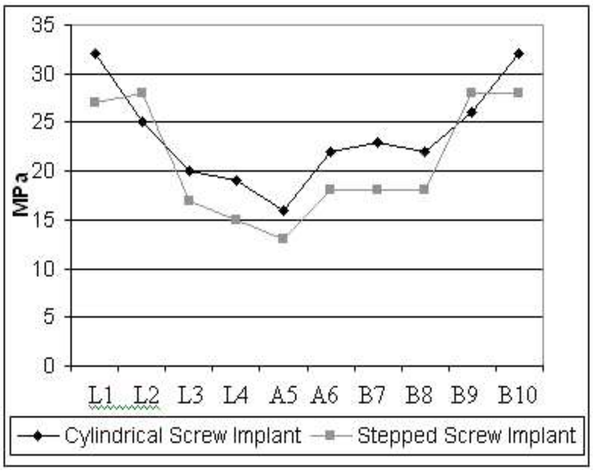


Fig. 5.
Interface between the implant and trabecular bone along path sections L1 to B10, of each step section of stepped screw implant and corresponding section of cylindrical screw implant.

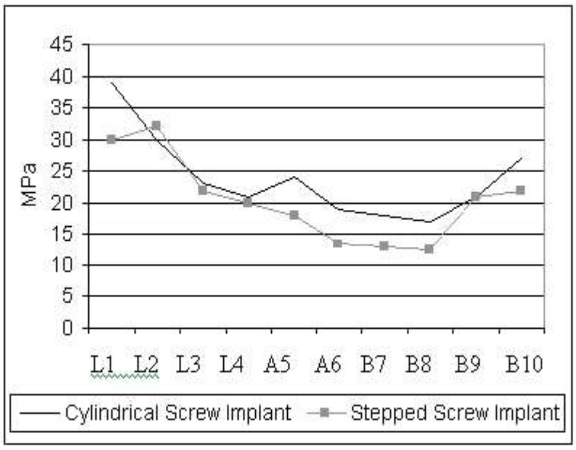
The stress distributions around the interface of implant neck and cortical bone were also similar. Furthermore, at the implant-trabecular bone interface of the stepped screw implant model, for all loading conditions, stresses were more evenly distributed, especially under the oblique loading condition. As shown in Fig. 6C, under oblique loading conditions, the maximum von Mises stress in the trabecular bone in the stepped screw implant model was 32 MPa and located on the notch between the first step section (L1, Fig. 5B) and the second step section (L2, Fig. 5B). This stress level was 17.9% lower than the cylindrical screw implant model, which was 39 MPa, on the inferior notch between the first thread and the second thread (L1, Fig. 5A). As shown in Fig. 6A, under the vertical load the stress difference between implant models was much smaller. The reason was that the absolute stress value was very small, less than 7 MPa under a vertical load of 100 N. As shown in Fig. 6B, under the horizontal load, the stress difference became obvious. The maximum stress difference was about 12.5%, i.e. 28 MPa in the stepped screw implant and 32 MPa in the cylindrical screw implant. However, the maximum stress difference was less than in the model under the oblique load.



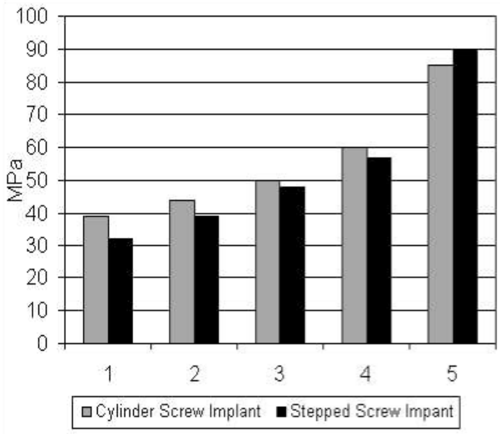
6A



6B



6C



7

Fig. 6. Maximum von Mises stress at the interface along path L1 to B10 between implant and trabecular bone under A, vertical load of 100 N, B, horizontal load of 100 N, and C, oblique load of 141 N at 45 degrees inclination (L: Lingual; A: Apex; B: Buccal).

Fig. 7. Maximum von Mises stress at the interface between the implant neck and the tranbecular bone under an oblique load of 141 N at 45 degrees inclination with different cortical bone properties condition 1 to 5.

Different cortical bone properties under oblique loading

The results shown in Fig. 7 indicated that cortical bone quality affected maximum von Mises stress in the trabecular bone of the implant-bone model. When decreased cortical bone quality was modeled (reducing the Young's modulus), the stress distribution changed significantly. When comparing the cylinder screw and the single stepped screw implants, it was found that in the cortical bone immediately adjacent to the implant neck, there were no significant differences in stress distribution. Furthermore, at the implant-trabecular bone interface of the stepped screw implant model, the maximum differences were found in the model where a Young's modulus of normal cortical bone (13,400MPa) was used. When decreasing the Young's modulus of the cortical bone from 13,400 MPa to 5,000 MPa, the maximum stresses in the trabecular bone of the stepped screw implant model were, respectively, about 17.9%, 11.4%, 4.0%, and 5.0% lower than their counterparts in the cylindrical implant model. This indicated that the model with normal cortical bone properties showed the highest stress difference in the models and the models with less than 10,000 MPa Young's modulus did not show obvious stress differences.

Different boundary conditions

Different boundary conditions under an oblique load and normal cortical bone condition were compared for the four single stepped screw implants. The maximum stress at the implant-cortical bone interface did not exhibit a significant difference with changing supporting constraints and thread profile configurations. A maximum of 20% stress difference in the trabecular bone-implant interface was observed between the four models with middle supporting constraints. However, when the supporting constraints were reduced from the middle to the base of the bone segment, the stress distributions were changed greatly and the thin-thread (T) model exhibited significantly higher stresses compared to the other three models. With the base supporting constraint the S1 model also exhibited a higher stress level than the other two thread profiles (V and S2). At the trabecular bone-implant interface, the stress level was approximately 30% higher.

Discussion

The results show that when comparing the force on the implant, the vertical and horizontal loads did not cause large differences in the stress distribution for the different implant-bone systems. For a vertical load condition, the absolute stress was very low, thus the stress difference was relatively less significant. A horizontal load induced higher absolute stresses but the relative stress difference was less than the one under the oblique load. The oblique load condition induced the largest difference in the stress distribution in the trabecular bone. The result confirmed a previous study concerning different implants and loading conditions.[13] In a pilot study, Holmgren et al.[9] found similar results in terms of the effect of loading direction on the stress distributions.

When the cortical bone quality was decreased to a level similar to trabecular bone, the stress distribution changed significantly. Although the maximum von Mises stress was lower in the stepped screw implant model compared to the cylindrical screw implant model, the stress difference between the two models also reduced with decreasing cortical bone quality. There was no significant stress difference between the two models when the Young's modulus of the cortical bone was lower than 10,000 MPa. When the Young's modulus was that of normal cortical bone (13,400 MPa), the difference in the maximum von Mises stress in the implant-trabecular bone interface area reached the highest level (13.4 MPa).

The stress difference in the cortical bone did not seem to be greatly influenced by how the mandible models were constrained. The maximum von Mises stress distribution showed that, in cortical bone-implant interfaces, there were no differences in any of the models. Therefore, when comparing various implant systems with similar implant neck and implant length, it is less important to study the stress level at the cortical bone interface caused by varying load inclination, cortical bone quality and supporting constraint positions. However, at the trabecular bone-implant interface, the stress distribution was greatly influenced by the supporting constraint positions. With middle supporting constraints, the thin-thread (T) model demonstrated the highest difference in maximum stress level among the other three models. However, with base supporting constraints, the T and S1 models gave higher maximum stresses than the V and S2 models. The results imply that v-thread (V) and large square-thread (S2) are the optimal thread profile for GJP stepped screw implants. Furthermore, minimal supporting constraints on the implant-bone model enabled a clearer differentiation of the stress distribution between the different stepped screw models at the trabecular bone-implant interface.

This study showed that an oblique load, high Young's modulus of the cortical bone, and minimal supporting constraints are important parameters in the implant design optimization. Other studies mentioned some other bone-implant parameters, however, most of them did not demonstrate obvious stress differences in a comparable way.[13,14,22,23] One report showed comparable stress differences but did not address other important parameters, such as oblique loading.[24] The authors' study has demonstrated that "model based optimization" could provide an effective way to reveal stress difference in the finite element model of a dental implant system.

Finally the authors would like to point out that although this study provided qualitative results to identify the key parameters for a model based implant design optimization, it was based on certain assumptions and simplified 2D models. Since the 2D model was built in the direction of principal loading of the tooth and the interaction load between neighbouring teeth is negligible, the method used here was sound and supported by previous studies. However, when the load interaction effect of neighbouring teeth is taken into account and an accurate analysis including the stiffness of the support structure is required, a 3D model should be used. Furthermore, when this method is used to investigate other bone-implant systems, such as maxillary bone-implants and orthopedic bone-implants a 3D model is recommended for biomechanical implant design optimization.

In summary, an oblique load, high Young's module modulus of the cortical bone, and minimal support constraints are important parameters. With these three substantial parameters a model-based optimization could be carried out effectively, helping the dentist to select a new implant system using FE analysis. Two substantial parameters (oblique load and cortical bone property) identified in this study provided biomechanical guidelines in clinical use of the implant system that will be subjected to immediate functional loads.

Acknowledgments

This work was partly supported by the National Natural Science Foundation of China (90406011), and Research and Development Program of Science & Technology in Zhejiang of China (2004C23009). The implant designs mentioned in this work were applied China Patent Nos. 200610052488.8 and 200620105708.4.

References

1. Branemark PI, Engstrand P, Ohrnell LO, Grondahl K, Nilsson P. Branemark Novum: a new treatment concept for rehabilitation of the edentulous mandible. Preliminary results from a prospective clinical follow-up study. Clin Implant Dent Relat Res 1999; 1: 2-16.
2. Romeo E, Chiapasco M, Ghisolfi M, Vogel G. Long-term clinical effectiveness of oral implants in the treatment of partial edentulism. Seven-year life table analysis of a prospective study with ITI dental implants system used for single-tooth restorations. Clin Oral Implants Res 2002; 13: 133-43.
3. Ferrigno N, Laureti M, Fanali S. A long-term follow-up study of non-submerged ITI implants in the treatment of totally edentulous jaws. Part I: Ten-year life table analysis of a prospective multicenter study with 1286 implants. Clin Oral Implants Res 2002; 13: 260-73.
4. Adell R, Lekholm U, Rockler B, Branemark PI. A 15-year study of osseointegrated implants in the treatment of the edentulous jaw. Int J Oral Surg 1981; 10: 387-416.
5. Adell R, Eriksson B, Lekholm U, Branemark PI, Jemt T. Long-term follow-up study of osseointegrated implants in the treatment of totally edentulous jaws. Int J Oral Maxillofac Implants 1990; 5: 347-59.
6. Ericsson I, Nilson H, Lindh T, Nilner K, Randow K. Immediate functional loading of Branemark single tooth implants. Clin Oral Implants Res 2000; 11; 26-33.
7. Geng JP, Tan KBC, Liu GR. Applications of finite element analysis in implant dentistry, a review of literatures. J Prosthet Dent 2001; 85: 585-98.
8. Bidez MW, Misch CE. Force transfer in implant dentistry: Basic concepts and principles. J Oral Implantology 1992; 18: 264-74.
9. Geng JP, Liu HC. Exceptional Prosthodontics. Hong Kong: Tranfor Press; 1999. p. 163-9.

10. 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 Implantology 1998; 24: 80-8.
11. 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: 633-40.
12. Holmes DC, Loftus JT. Influence of bone quality on stress distribution for endosseous implants. J Oral Implantology 1997; 23: 104-11.
13. Rieger MR, Fareed K, Adams WK, Tanquist RA. Bone stress distribution for three endosseous implants. J Prosthet Dent 1989; 61: 223-8.
14. Rieger MR, Adams WK, Kinzel GL. Finite element survey of eleven endosseous implants. J Prosthet Dent 1990; 63: 457-65.
15. Valen M. The relationship between endosteal implant design and function: maximum stress distribution with computer-formed three-dimentional Flexi-cup blades. J Oral Implantology 1983; 11: 49-71.
16. Patra A, DePaolo JM, D'Souza KS, DeTolla D, Meenaghan ME. Guidelines for analysis and redesigh of dental implants. Implant Dent 1998; 7: 355-66.
17. DeTolla DH, Andreana S, Patra A, Buhite R, Comella B. The role of the finite element model in dental implants. J Oral Implantology 2000; 16: 77-81.
18. Geng JP, Ma Q, Xu W, Tan KBC, Liu GR. Finite element analysis of four thread-form configurations in a stepped screw implant. J Oral Rehabil 2004; 37: 233-9.
19. Cowin SC. Bone Mechanics. Boca Raton: CRC Press; 1989.
20. Cook, SD, Klawitter, JJ, Weinstein, AM. A model for the implant-bone interface characteristics of porous dental implants. J Dent Res 1982; 61: 1006-9.
21. Borchers, L, Reichart, P. Three-dimensional stress distribution around a dental implant at different stages of interface development. J Dent Res 1983; 62: 155-9.
22. Van Oosterwyck H, Duyck J, Vander Sloten J, et al. The influence of bone mechanical properties and implant fixation upon bone loading around oral implants. Clin Oral Implants Res 1998; 9: 407-18.
23. Pilliar RM, Deporter DA, Watson PA, Valiquette N. Dental implant design-Effect on bone remodelling. J Biomed Mater Res 1991; 25: 467-83.
24. Eskitascioglu G, Usumez A, Sevimay M, Soykan E, Unsal E. The influence of occlusal loading location on stresses transferred to implant-supported prostheses and supporting bone: A three-dimensional finite element study. J Prosthet Dent 2004; 91: 144-50.

Correspondence to:

Dr. Wei Xu
Centre for Biomedical Engineering, School of Engineering, University of Surrey
Guildford, Surrey, GU2 7XH, United Kingdom
Fax: +44-1483-306039 E-mail: W.Xu@surrey.ac.uk

Received September 13, 2005. Accepted February 12, 2007.