Clin Oral Invest (2010) 14:411–416
DOI 10.1007/s00784-009-0305-1

ORIGINAL ARTICLE

The effect of thread design on stress distribution in a solid screw implant: a 3D finite element analysis

Oğuz Eraslan · Özgür İnan

Received: 16 December 2008 / Accepted: 8 June 2009 / Published online: 20 June 2009

Abstract The biomechanical behavior of implant thread plays an important role on stresses at implant-bone interface. Information about the effect of different thread profiles upon the bone stresses is limited. The purpose of this study was to evaluate the effects of different implant thread designs on stress distribution characteristics at supporting structures. In this study, three-dimensional (3D) finite element (FE) stress-analysis method was used. Four types of 3D mathematical models simulating four different thread-form configurations for a solid screw implant was prepared with supporting bone structure. V-thread (1), buttress (2), reverse buttress (3), and square thread designs were simulated. A 100-N static axial occlusal load was applied to occlusal surface of abutment to calculate the stress distributions. Solidworks/Cosmosworks structural analysis programs were used for FE modeling/analysis. The analysis of the von Mises stress values revealed that maximum stress concentrations were located at loading areas of implant abutments and cervical cortical bone regions for all models. Stress concentration at cortical bone (18.3 MPa) was higher than spongious bone (13.3 MPa), and concentration of first thread (18 MPa) was higher than other threads (13.3 MPa). It was seen that, while the von Mises stress distribution patterns at different implant thread models were similar, the concentration of compressive stresses were different. The present study showed that the use of different thread form designs did not affect the von Mises concentration at supporting bone structure. However, the compressive stress concentrations differ by various thread profiles.

O. Eraslan (✉) · Ö. İnan
Department of Prosthodontics, Faculty of Dentistry,
University of Selcuk,
42079 Kampus,
Konya, Turkey
e-mail: oguzeraslan@selcuk.edu.tr

Keywords Implant thread form design · Dental implants · Biomechanics · Finite element analysis · Stress distribution

Introduction

Dental implants function to transfer load to surrounding biological tissues [12]. Thus, the primary functional design objective is to manage (dissipate and distribute) biomechanical loads to optimize the implant-supported prosthesis function [12]. Implant thread configuration is an important objective in biomechanical optimization of dental implants [2, 7, 15, 20, 21]. The implant-bone interface can be easily compromised by high stress concentrations that are not dissipated through the implant configuration [8]. It is necessary that biomechanical concepts and principles are applied to thread design of dental implant to further enhance clinical success [8].

Thread geometry includes thread pitch, depth, and shape [7, 12]. Although thread pitch and depth could affect the stress distribution, traditionally, the manufacturers have provided implant system a constant pitch and depth [8]. So, for commercial implant system, a better design of thread configuration is emphasized [8].

Threads are designed to maximize initial contact, enhance surface area, and facilitate dissipation of stresses at the bone-implant interface [10]. Thread shapes in dental implant designs include square, V-shape, and buttress [12]. In conventional engineering applications, the V-thread design is called a “fixture” and is primarily used for fixating metal parts together, not load transfer [17]. The buttress thread shape was designed initially for and is optimized for pullout loads [12]. The square thread provides an optimized surface area for intrusive, compressive load transmission [12].

A key factor for the success or failure of a dental implant is the manner in which stresses are transferred to the surrounding bone [22]. Shear loading is reported as the most detrimental loading profile for bone [12]. The reduction in shear loading at the thread-to-bone interface provides for more compressive load transfer, which is particularly important in compromised D3 and D4 bone [12]. The finite element analysis (FEA) allows researchers to predict stress distribution in the contact area of implants with cortical bone and around the apex of implants in trabecular bone [8] without the risk and expense of implantation [5]. FEA is utilized at current study to evaluate the effects of different implant thread designs on stress distribution characteristics at supporting structures. The null hypothesis was that different implant thread shape does not affect the stress distribution within supporting structures.

Materials and methods

The study was conducted using a 3D FE method and the Solidworks 2007 9.0.3 structural analysis program (Solidworks Corporation, Concord, MA, USA). An ITI solid cylindrical screw implant, 3.8-mm diameter, 10-mm bone sink depth (Straumann AG, Waldenburg, Switzerland) was modeled. The implant system was modeled as a single body unit with its abutment. Cortical and cancellous bones are also modeled representing the cross-section of the posterior human mandible. Two-millimeter thickness of cortical bone was modeled around the cancellous bone and implant neck. The 3D solid screw implants were modeled with similar conditions of same thread number, position, height (D), and pitch (P) (Fig. 1). Initially, cross-sections of structures included in mathematical model were sketched at front and right planes separately for each unit at computer environment. Coordinates of the contouring points were then entered as border nodes of mathematical models. These nodes were joined to form each structures 3D volume that together defined the final geometry of FE model.

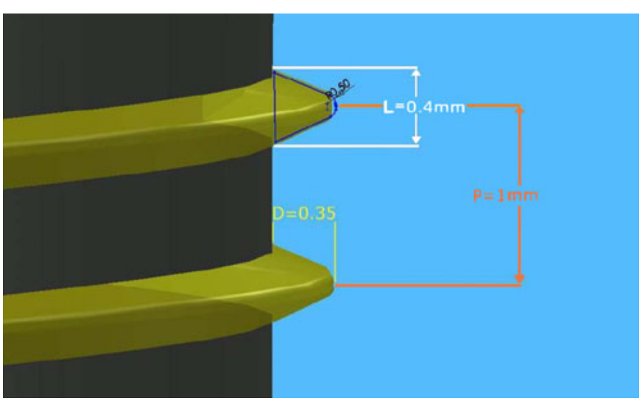

Fig. 1 Schematic illustration of thread profile properties used in study. Threads pitch (*P*), height or length (*L*), depth (*D*)

In order to better understand the stress distribution, four different thread-form configurations are compared: V-thread, buttress, reverse buttress, and square shape thread forms of 0.4 mm thread width (L; Fig. 2). The geometric models were meshed with tetrahedral quadratic elements (Fig. 3). Each mathematical model included approximately 186,000 nodes and 133,000 solid elements. The bottom exterior nodes of the alveolar bone in the FEM models were fixed in all directions as the boundary condition (Fig. 3). A 100-N static axial occlusal load was applied to occlusal surface of abutment to calculate the stress distributions (Fig. 3).

Materials used in study were assumed to be homogenous and isotropic. Elastic properties of materials (Young's modulus (E) and Poisson's ratio (μ)) were determined from the literature and given in Table 1. The FE modeling was accomplished with the Solidworks software program, and analyses were run at Cosmosworks software program, which is integrated with Solidworks.

Results

Results were presented by considering von Mises criteria [1, 13, 18, 19, 25], and principal compressive stress field. Calculated numerical data were transformed into color graphics to better visualize mechanical phenomena in the models. Both 3D whole model view and mesiodistal cross-sectional views were presented for each thread type. All stress values were indicated in mega pascals (MPa).

The analysis of the von Mises stress values revealed that maximum stress concentrations were located at loading areas of implant abutments for all models (Fig. 4). Also, high stress values were located at cervical cortical bone regions adjacent to implants at all models. It can be seen that the stress concentration at cortical bone structure was higher than that of spongious bone at 3D whole model view (Fig. 4).

When the mesiodistal cross-sectional views of bone structures (Fig. 5) were evaluated, it was seen that the stress distribution patterns at different implant thread models were similar. However, the maximum von Mises stress values at different models were not similar; 17.7 MPa for V-thread, 18.3 MPa for buttress thread, 18.2 MPa for reverse buttress thread, and 17.3 MPa for square thread type. These maximum values were observed at cervical cortical bone regions adjacent to implants and at bone structure adjacent to first thread which was located at cortical bone structure. Also, from this cross-section view, it was clearly seen that

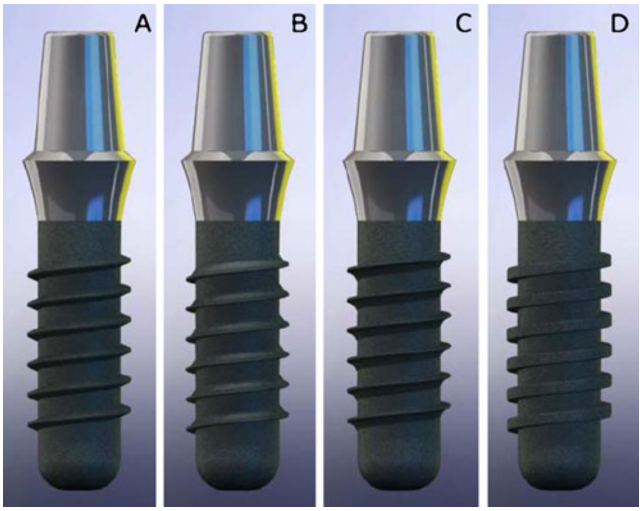

Fig. 2 Solid screw implants modeled in study that has four different thread profiles. V-thread (**a**), buttress (**b**), reverse buttress (**c**), and square shape (**d**) threads

stress concentration at cortical bone structure (17.3–18.3 MPa) was higher than that of spongious bone (11.7–13.3 MPa). Also, the stress concentration of bone structure adjacent to first thread which was located at cortical bone structure was higher than that of other threads which were located at spongious bone.

When the compressive stress values around the implants were evaluated (Fig. 6), maximum compressive stress value was 18 MPa, and this value was observed at all four thread types. Nevertheless, the intensity (area covered) of this maximum stress value was different among the different thread types. The maximum stress intensity area covered the biggest area at reverse buttress thread type. It was similar at buttress and square thread type, and it was lower than reverse buttress thread type. The minimum stress intensity area was observed with V-thread shape.

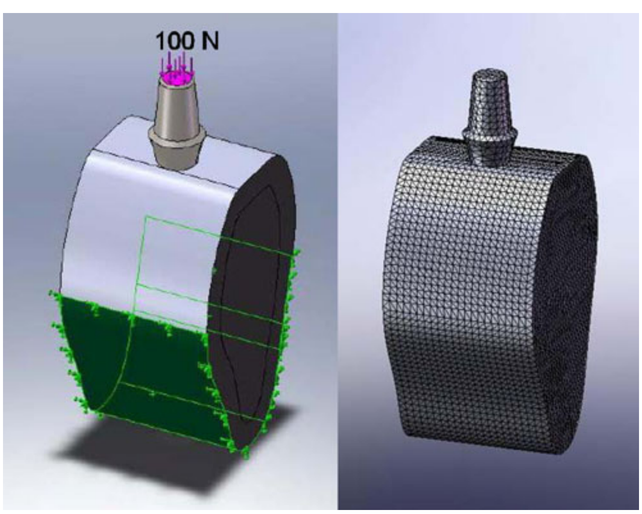

Fig. 3 Illustration of 3D FE model, load application, and boundary condition. *Pink arrow* represents the load application, and *green area* is assumed as fixed as boundary condition

Table 1 Mechanical properties of investigated materials

Material	Elastic modulus (E; GPa)	Poisson's ratio (μ)
Titanium [16]	110	0.35
Cortical bone [23]	13.7	0.30
Cancellous bone [23]	1.37	0.30

Discussion

This FEA-utilized study showed that while the von Mises stress distribution characteristics at supporting structures were similar at four different implant thread designs, the distribution of compressive stresses were different, and these results were similar to a study [8] that compared the square and V-thread types and stated that thread form configurations does not greatly affect the stress distribution in cortical bone. Current study adds the buttress and reverse buttress thread shapes to this findings. Based on these results, the null hypothesis that the thread shape would not affect the stress distribution at supporting bone structure was rejected for compressive stresses and accepted for von Mises stresses.

It was reported in the literature [4, 12] that stress (compressive) was more evenly distributed in the case when the implant thread shape was square than V-thread shape. Findings of current study were in accordance with them; however, current study adds buttress and reverse buttress thread type to comparison and especially the reverse buttress thread shape seems to be more advantageous if compressive types of stresses are desired to occur at thread-bone structure interface. In contrast, Hansson and Werke [9] reported that the thread profile has a profound effect upon the magnitude of stresses in the bone; in the present study, stress distribution patterns at different implant thread models were similar. The reason for this opposition could be that they assumed the implant to be infinitely long and embedded in a cortical bone cylinder. Also, that implant was not having neck-abutment parts, and implants were assumed rotationally symmetric. At current study, threads were modeled with revolution downwards as it is commercially available in the market.

Previous researches reported that the stress was concentrated in the cervical cortical bone region [6] and the highest stress concentration occurs at the region in jaw bone adjacent to the first thread of implant. Current FE study confirmed that the stress was concentrated at the cervical region and first thread. It has been suggested that compressive stresses may act as a bone maintenance

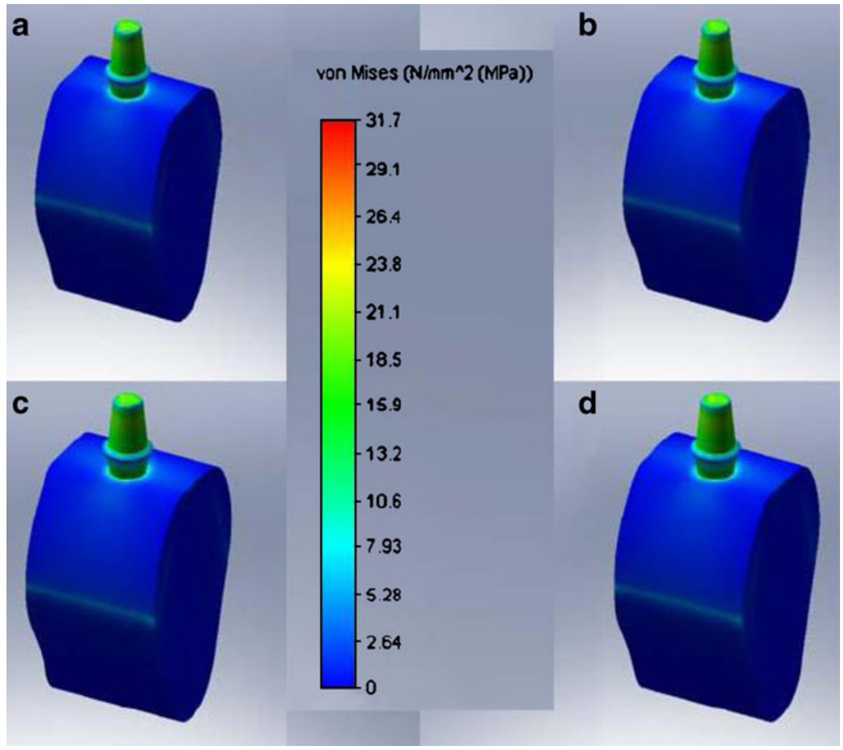

Fig. 4 Distribution of von Mises stresses (MPa) at main models. V-thread (**a**), buttress (**b**), reverse buttress (**c**), and square shape (**d**) threads. *Blue to red colors* represents stress values from lower to higher, respectively

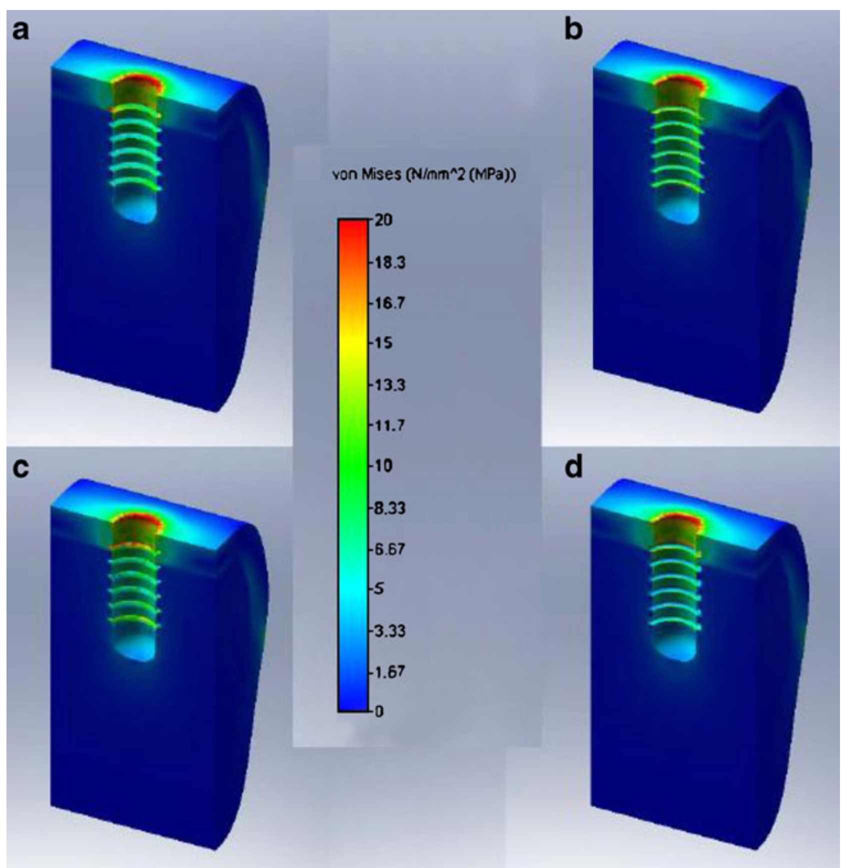

Fig. 5 Distribution of von Mises stresses (MPa) at mesio-distal cross-section view of bone structure. V-thread (**a**), buttress (**b**), reverse buttress (**c**), and square shape (**d**) threads. *Blue to red colors* represents stress values from lower to higher, respectively

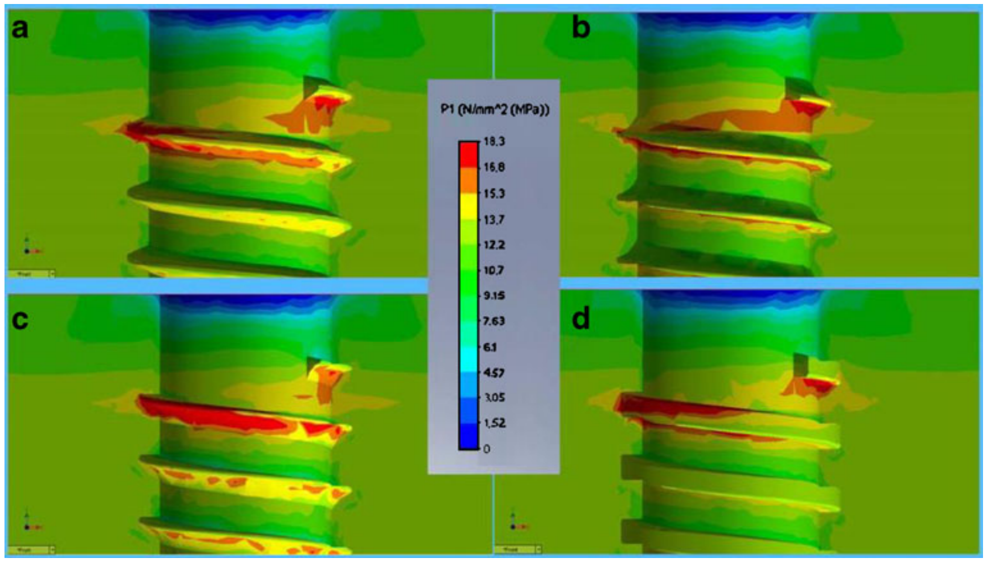

Fig. 6 Distribution of compressive stresses (MPa) at mesiodistal cross-section, zoomed view of bone structure adjacent to cervical implant region. V-thread (**a**), buttress (**b**), reverse buttress (**c**), and square shape (**d**) threads. *Blue to red colors* represents stress values from lower to higher, respectively

stimulus [12]. Also, high shear stress concentration at implant-bone interface [12] and insufficient mechanical stimulation (compressive forces) have been suggested to be a major etiological factor [24] behind the marginal bone loss often observed at dental implants. At current FE study, high von Mises stress concentrations and lack of compressive stresses were observed at cervical bone region. Thus, thread designs like reverse buttress, which form more compressive stresses, may be considered for bone stimulation.

As mentioned before, thread pitch and depth also could affect the stress distribution [8]. This topic was analyzed in literature and effects of thread pitch, depth, and width were reported [9, 11]. Also, it was noticed that a systematic analysis of the effect of different thread profiles upon the bone stresses has not yet been made [9]. Thus, it was aimed to compare four different thread profiles at current study. However, as a limitation of current study, analyzed implants did not have threads at neck region. Implants that have threads at neck region are commercially available, and this configuration type can be compared at a further study.

The size of occlusal force is selected as 100-N value. However, it is not necessary for this force to match the reality exactly because standardization between conditions has been ensured in the current study, and the conditions have been compared qualitatively with each other. Chen and Xu [3] have emphasized that the value of FEM modeling is in relative values calculated at distribution pattern.

The FEM results are presented as stresses distributed in the investigated structures. These stresses may occur as tensile, compressive, shear, or a stress combination known as equivalent von Mises stresses. Von Mises stresses depend on the entire stress field and are a widely used indicator of the possibility of damage occurrence [13, 14]. Thus, von Mises and compressive stresses were chosen for presentation of results.

As with many in vitro studies, it is difficult to extrapolate the results of this study directly to a clinical situation. The model used in this study implied several assumptions regarding the simulated structures. The structures in the model were all assumed to be homogeneous, isotropic, and to possess linear elasticity. The properties of the materials modeled in this study, particularly the living tissues, however, are different. Also, it is important to point out that the stress distribution patterns may have been different depending on the materials and properties assigned to each layer of the model and the model used in the experiments. Thus, the inherent limitations in this study should be considered. Further studies that better simulate the oral environment and including fatigue loading are recommended.

Conclusion

Within the limitations of this study, the following conclusions were drawn:

1. The different implant thread forms do not affect the von Mises stress distributions at supporting bone structure.
2. Different implant thread forms produce different compressive stress intensities at bone structure.
3. Cortical bone and bone structure adjacent to first thread bears more both von Mises and compressive stresses than spongious bone.

Conflict of interest This study is funded by the Research Projects Counsil of the University of Selcuk. The authors declare that they

have no financial, professional, or other personal interest that could influence the position presented in the paper.

References

1. Beer FP, DeWolf JT, Johnston ER (2005) Mechanics of materials, 4th edn. McGraw-Hill International, Singapore, pp 360–378
2. Brunski JB (1999) In vivo bone response to biomechanical loading at the bone/dental–implant interface. Adv Dent Res 13:99
3. Chen J, Xu L (1994) A finite element analysis of the human temporomandibular joint. J Biomech Eng 116:401–407
4. Chun HJ, Cheong SY, Han JH, Heo SJ, Chung JP, Rhyu IC, Choi YC, Baik HK, Ku Y, Kim MH (2002) Evaluation of design parameters of osseointegrated dental implants using finite element analysis. J Oral Rehabil 29:565–574
5. Cook SD, Klawitter JJ, Weinstein AM (1982) A model for the implant-bone interface characteristics of porous dental implants. J Dent Res 61:1006–1009
6. Cruz M, Wassall T, Toledo EM, Barra LP, Lemonge AC (2003) Three-dimensional finite element stress analysis of a cuneiform-geometry implant. Int J Oral Maxillofac Implants 18:675–684
7. Geng JP, Ma XX (1995) A differential mathematical model to evaluate side-surface of an Archimede implant. Shanghai Shengwu Gongcheng Yixue 50:19
8. Geng JP, Ma QS, XU W, Tan KBC, Liu GR (2004) Finite element analysis of four thread-form configurations in a stepped screw implant. J Oral Rehabil 31:233–239
9. Hansson S, Werke M (2003) The implant thread as a retention element in cortical bone: the effect of thread size and thread profile: a finite element study. J Biomech 36:1247–1258
10. Ivanoff CH, Grönhahl K, Sennerby L, Bergström C, Lekholm U (1999) Influence of variations in implant diameters: a 3- to 5-year retrospective clinical report. Int J Oral Maxillofac Implants 14:173–180
11. Kong L, Hu K, Li D, Song Y, Yang J, Wu Z, Liu B (2008) Evaluation of the cylinder implant thread height and width: a 3-dimensional finite element analysis. Int J Oral Maxillofac Implants 23:65–74
12. Misch CE (2005) Dental implant prosthetics. Mosby, St Louis, pp 322–347
13. Pegoretti A, Fambri L, Zappini G, Bianchetti M (2002) Finite element analysis of a glass fibre reinforced composite endodontic post. Biomaterials 23:2667–2682
14. Pierrisnard L, Bohin F, Renault P, Barquins M (2002) Corono-radicular reconstruction of pulpless teeth: a mechanical study using finite element analysis. J Prosthet Dent 88:442–448
15. Rieger MR, Adams WK, Kinzel GL (1990) Finite element survey of eleven endosseous implants. J Prosthet Dent 63:457
16. Sevimay M, Usumez A, Eskitascioglu G (2005) The influence of various occlusal materials on stresses transferred to implant-supported prostheses and supporting bone: a three-dimensional finite-element study. J Biomed Mater Res B Appl Biomater 73:140–147
17. Singley JE, Mischke CR (1989) Mechanical engineering design, 5th edn. McGraw-Hill, New York, pp 123–157
18. Timoshenko S, Young DH (1968) Elements of strength of materials, 5th edn. Wadsworth, Florence, p 377
19. Ugural AC, Fenster SK (2003) Advanced strength and applied elasticity, 4th edn. Prentice-Hall, New York, pp 155–157
20. Valen M (1983) The relationship between endosteal implant design and function: maximum stress distribution with computerformed three-dimensional Flexi-cup blades. J Oral Implantol 11:49
21. Valen M, Locante WM (2000) LaminOss immediate-load implants: 1. Introducing osteocompression in dentistry. J Oral Implantol 26:177
22. Van Oosterwyck H, Duyck J, Vander S, Van der Perre G, De Cooman M, Lievens S, Puers R, Naert I (1998) The influence of bone mechanical properties and implant fixation upon bone loading around oral implants. Clin Oral Implants Res 9:407
23. Weinstein AM, Klawitter JJ, Cook SD (1980) Implant-bone interface characteristic of bioglass dental implants. J Biomed Mater Res 14:23–29
24. Wiskott HW, Belser UC (1999) Lack of integration of smooth titanium surfaces: a working hypothesis based on strains generated in the surrounding bone. Clin Oral Implants Res 10:429–444
25. Yang HS, Lang LA, Molina A, Felton DA (2001) The effects of dowel design and load direction on dowel-and-core restorations. J Prosthet Dent 85:558–567