Stomatologija, Baltic Dental and Maxillofacial Journal, 11: 55-59, 2009

Stress distribution of an internal connection implant prostheses set: A 3D finite element analysis

Regenio Mahfuz Herbstrith Segundo, Hugo Mitsuo Silva Oshima, Isaac Newton Lima da Silva, Luis Henrique Burnett Junior, Eduardo Goncalves Mota, Liangrid Lutiani Silva

SUMMARY

Objective. This study evaluated the stress distribution by loads, under a internal connection implant sytem with two sort of abutment screws and prosthetic crown models at ten observation points.

Materials and method: The analysis were made in two models with internal butt joint, and with gold and titanium screw, respectively. The load was 382N with 90º to the occlusal surface and 15º to the implant axis at 4 and 6 mm from the implant center.

Results: In both models, a large amount of stress was located around the implant neck and little stress was concentrated along the abutment screw.

Conclusion: The simulations made suggest that the internal connection protects the abutment screw from the accumulated stresses; however, it exposes the implant walls to these stresses.

Key words: finite element analysis, abutment screw, biomechanic.

INTRODUCTION

Initially, titanium implants were used to rehabilitate completely edentulous patients, referred to as oral invalids, with the goal of reestablishing masticatory function. With the development of techniques and materials used in implant dentistry, other indications were added, and now implants are used to rehabilitate cases of partial and single edentulism [1,2] , combining function and esthetics.

In time and with the increasing use osseointegrated implants, a number of biomechanics problems began to appear [3]. In an endeavor to minimize such problems, alternative connection systems, based on the internal opposition of implants and abutments walls were developed, either a cone-shaped connection with an angle ranging from 8 to 11° [4,5], or one with an internal hexagon-like format [6,7]. Hypothetically it could reduce the stress on the abutment screw, because the oclusal stress transferred to the abutment could be divided between the internal joint walls and the abutment screw [7].

However, these changes diminished these biomechanical problems, but did not eliminate them, therefore, further longitudinal studies are required to make the use of these new technologies safer. Many computer and engineering scientists have used virtual models and environments, called finite element analyses, to run simulations and progressively test the resistance and stress distribution of tools and parts of machines used in present day life. According to Geng *et al*. (2001) [8], mechanical problems are solved by splitting the main problem into small and simple elements, arranged in a net shape, called a mesh, in which the analyzed variables are interpolated in the form mathematical functions.

The aim of this preliminary study was to make a qualitative comparison and evaluation, by means of 3D-finite element analysis, the stress distribution produced by simulated load under a internal connection implant sytem with two sort of abutment screws and prosthetic crown models at ten observation points.

*Pontificia Universidade Catolica do Rio Grande do Sul, Brazil

Regenio Mahfuz Herbstrith Segundo* – *DDS, PhD student*
Hugo Mitsuo Silva Oshima* – *DDS, PhD, assist. professor, Department of Prosthodontics*
Isaac Newton Lima da Silva* – *DDS, PhD, assist. professor, Department of Mechanic Engineering*
Luis Henrique Burnett Junior* – *DDS, PhD, assist. professor, Department of Dentistry*
Eduardo Goncalves Mota* – *DDS, assist. professor, Department of Dental Materials, Faculty of Dentistry*
Liangrid Lutiani Silva* – *Graduate program student*

Address correspondence to Regenio Mahfuz Herbstrith Segundo, Pontificia Universidade Catolica do Rio Grande do Sul, Guadalupe 90/601, Jardim Lindoia, Porto Alegre, RS, Brazil, 91050 - 250.
E-mail address: regesegundo@yahoo.com.br

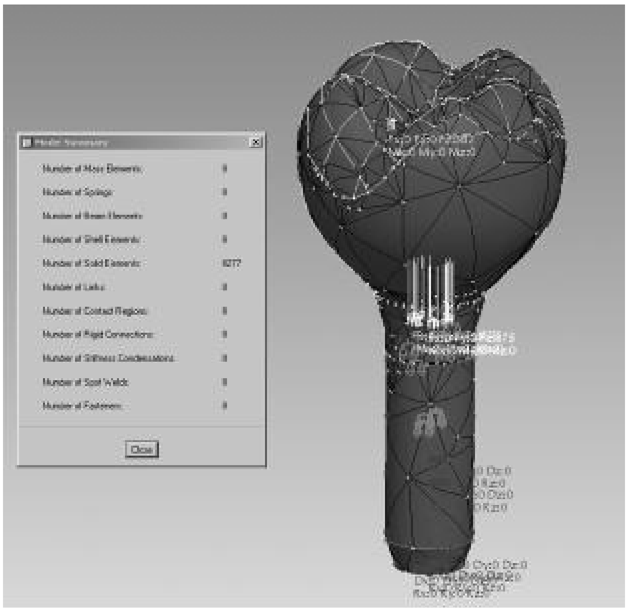

Fig. 1. Schematic image of model mesh configurations

MATERIALS AND METHOD

In this study, the following components and materials were used: 02 Osseotite Certain implants measuring 11.5x4.0 mm in size (3i, Implant Innovations, Palm Beach, Florida, USA), 02 Gengihue abutments of 2 mm (3i, Implant Innovations, Palm Beach, Florida, USA), 02 gold screws, Gold Tite® hexagonal head (3i, Implant Innovations, Palm Beach, Florida, USA). By exposing the test sets to a Profile Projector (Nikon model v.16, serial number 36914, RBC Certificate 7463.1) it was possible to develop two 3D virtual models.

Model 1 and 2 with pre-load of 20 N/cm^2 , applied on the contact surface of the screw head with the respective abutment, consisting of: model 1; (Osseotite Certain implant measuring 11.5 × 4.0 mm in size, Gengihue abutment of 2 mm, hexagonal gold abutment screw, Ag-Pd metal coping); model 2; (Osseotite Certain implant measuring 11.5 × 4.0 mm in size, Gengihue abutment of 2 mm, hexagonal titanium abutment screw, Ag-Pd metal coping) by means of the CAD computational tool (PRO-ENGINEER, PTC, Needham, MA, USA). The coronal end of the models was extracted from a real dental element, through Computer Tomography of a human tooth (46 dental element) due to the geometric complexity of the occlusal surface and imported to the PRO-E System. Models 1 and 2 were divided into reduced solid elements, originating 8277 tetrahedral elements each, giving rise to a virtual net called mesh (Figure 1). The intrinsic material characteristics used to construct the models are listed in Table 1. Next, virtual simulations of load applications on the models were done by the PRO-MECHANIC System, also developed by PTC, analyzing and comparing the stress distribution induced by the loads applied on the model structure by Von Misses Stress (EQV-MPa), the stresses were analyzed at ten different observation points (Table 2). A static load of 382N was applied, simulating loads in the molar region [9] and was applied parallel to and at a 15º inclination to the implant axis [10] in two different regions of the superior surfaces of metal ceramic crown models, at 4 mm and 6 mm from the center of the implant model [11]. To enhance evaluation of the results, some restrictions were placed on the implant model movements, simulating a vertical bone loss level of 3 mm from the implant platform.

RESULTS

Comparing the two types of screws, gold and titanium, in models 1 and 2, a clear balance was shown in the stress values of the ten observations points (Figure 2). This balance is stronger when direct comparison was made between the abutment screw points (M3'c, M4c, M6c, M7c, M8c, M9c). In these models (1 and 2) there was no linear behavior related to the load application point (Tables 3 and 4). In both models, a large amount of stress was located around the implant neck (Figures 3 and 4) and little stress was concentrated along the abutment screw (Figures 5 and 6).

Table 1. Characteristics of the materials used for simulating the models

	Models	Modulus of elasticity (mpa)	Poisson's ratio	References
Ti(ASTM-F67)	screws, abutments	100 x 10^3	0.34	Akour *et al* (2005)[24]
Ti6Al4V(ASTM-F-136)	Implants	110 x 10^3	0.34	Akour *et al* (2005)[24]
Gold alloy-type 3	screws	100 x 10^3	0.30	Geng *et al* (2001)[8]
Feldspatic ceramic	Esthetic material	68.9 x 10^3	0.28	Geng *et al* (2001)[8]
Ag-Pd Alloy	Coping	95 x 10^3	0.33	Geng *et al* (2001)[8]

Table 2. Points of observation – models 1 and 2

M0c	Abutment/Implant intersection – load application side
M1c	Abutment/Implant intersection – side opposite to load
M2c	Internal point of hexagon – load application side
M3c	Internal point of hexagon – side opposite to load
M3'c	Half of internal diameter of 1st thread of screw on side opposite to load.
M4c	Half of internal diameter of 1st thread of screw – on load application side.
M6c	2nd thread of screw – load application side
M7c	2nd threat of screw – side opposite to load
M8c	Half of internal diameter of 2nd thread of screw – on load application side
M9c	Half of internal diameter of 2nd thread of screw – on side opposite to load.

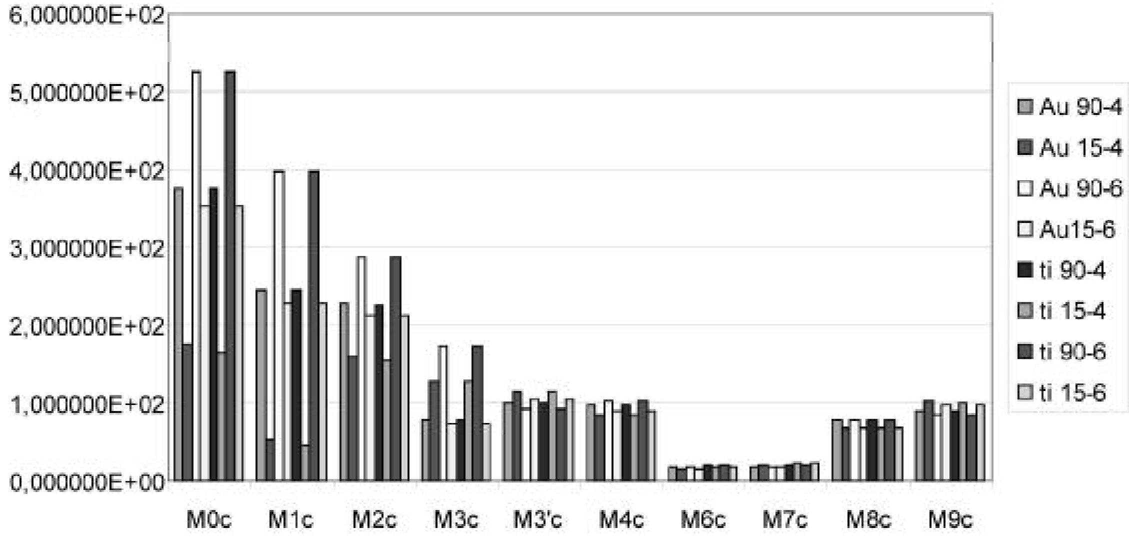

Fig. 2. Stresses x points of observation, Model 1 × Model 2

DISCUSSION

The biomechanics of the implant-screw-abutment complex is very differentiated. As the concepts of osseointegration began to consolidate, this region of the implant-prosthesis set started arousing interest, because the mechanical problems, such as loosening of prosthetic abutment screws, their fractures, and sometimes implant fractures that had long intrigued clinicians, began to occur more frequently, when compared with problems related to osseointegration. According to Binon *et al.* (1994), these problems might be caused, mainly due to inadequate torque, prostheses lacking adaptation and passive fit [12] , occlusal overload and unsuitable retainer screw design [13]. According to Wiskott et al. (2004) there is a directly proportional relationship between the preload applied on abutment screws and their resistance to fatigue, which may cause severe mechanical problems [14].

In the assessment made by Goodacre *et al.* (2003) through a systematic literature review, a large number of mechanical complications have been reported recently, and of the articles reviewed by the author, 1% of them showed the presence of fistula at the level of the prosthetic connection, 6% indicated loss of abutment screws, 45% of these losses being in unit crowns; abutment screw fractures occurred in 2% of the studies and implant fractures in 1% [15].

The use of the finite element method to analyze stress concentrations was initially introduced into implant dentistry by Weinstein *et al.* (1976) [16]. The models used can be bidimensional [11, 16] or three dimensional [17, 18]. The analyses can be done from the bone-implant interface point of view, relating stress concentrations and displacements between titanium and bone [19, 20, 21, 22, 23] in the implant-screw-abutment set relating the geometric form of the prosthetic connections and screw materials with potential risks for failure [24, 25] or suggesting modifications in im-

Table 3. Results of stress concentrations (MPa) at points of observation in Model 1

	Au90-4	Au15 -4	Au90- 6	Au15 -6
M0c	3.745875E+02	1.761721E+02	5.256543E+02	3.520527E+02
M1c	2.442097E+02	5.210352E+01	3.969740E+02	2.275380E+02
M2c	2.264654E+02	1.595427E+02	2.886220E+02	2.120026E+02
M3c	7.796789E+01	1.263894E+02	1.731918E+02	7.184100E+01
M3'c	9.923599E+01	1.146352E+02	9.148182E+01	1.047447E+02
M4c	9.707256E+01	8.483411E+01	1.017304E+02	8.980682E+01
M6c	1.777277E+01	1.587256E+01	1.752911E+01	1.548951E+01
M7c	1.722766E+01	1.891891E+01	1.716692E+01	1.870405E+01
M8c	7.681787E+01	6.810822E+01	7.745234E+01	6.848799E+01
M9c	8.960729E+01	1.015408E+02	8.589566E+01	9.646799E+01

Table 4. Results of stress concentrations (MPa) at points of observation in Model 2

	ti 90-4	ti 15-4	ti 90-6	ti 15-6
M0c	3.745000E+02	1.641465E+02	5.255544E+02	3.519912E+02
M1c	2.442249E+02	4.399045E+01	3.970392E+02	2.275662E+02
M2c	2.261010E+02	1.546018E+02	2.884694E+02	2.118834E+02
M3c	7.803716E+01	1.270128E+02	1.732793E+02	7.148720E+01
M3'c	9.908669E+01	1.142111E+02	9.157460E+01	1.047777E+02
M4c	9.732100E+01	8.415629E+01	1.020215E+02	9.003782E+01
M6c	2.069690E+01	1.839425E+01	2.043919E+01	1.808135E+01
M7c	1.963538E+01	2.147903E+01	1.951370E+01	2.129763E+01
M8c	7.644028E+01	6.721249E+01	7.717117E+01	6.821058E+01
M9c	8.940670E+01	1.009422E+02	8.582062E+01	9.636969E+01

Fig. 3. Schematic image of stress distribution on implant in Model 1, with load applied at an angle of 90° at 6mm from the center of the implant

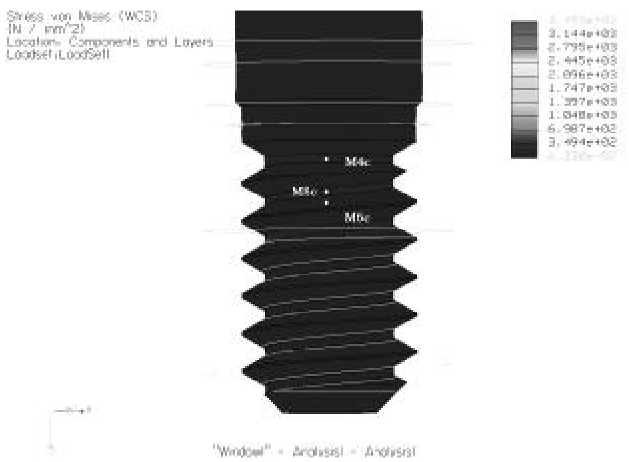

Fig. 4. Schematic image of stress distribution on abutment screw in Model 1, with load applied at an angle of 90° at 6mm from the center of the implant

plant and prosthetic component designs to maximize clinical performance [26, 27].

The majority of the studies related to *FEA* in the implant connection – prosthesis used loads of magnitude ranging between 35N 25] and 80N [24], which justify the high values found in this study, since the load used was 382N, around 4 times the value of the highest load found in the literature. Although laboratory and clinical studies [3, 5, 28, 14, 29] used cyclic loads related to the causes of loss of abutment screws with dynamic fatigue, the majority of studies revised, which used FEA in their methodology, used static loads to facilitate stress analysis, since more complex virtual tests require materials considered anisotropic, non linear and heterogeneous [8].

The highest and lowest accumulated stress values in models 1 and 2 were: 5.256543E+02 MPa, 5.255544E+02 MPa and 1.548951E+01 MPa, 1.808135E+01 MPa, respectively. According to Jörnéus *et al*. (1992), the only screw capable of maintaining a single implant supported restoration stable under extreme load situations, is the gold screw because of the mechanical characteristics of the alloy [30].

Models 1 and 2 demonstrated very closed values when homologous situations were compared, both at the points situated on the screw and those located outside of it. At some points, the stresses found were practically equal, such as for example, at the point of observation M1c, at the half of the internal hexagon face, for which the stress values were 2.442097E+02 MPa for model 1 and 2.442249E+02 MPa for model 2, suggesting that for implant models studied in this paper, the material the abutment screw is made of does not have great influence on the stress distribution along the implant-abutment complex. This can be explained due to the division of the masticatory forces transmitted between the abutment screw and the walls of the

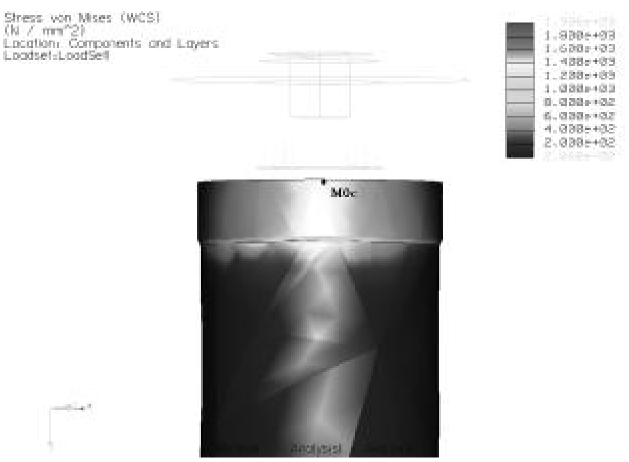

Fig. 5. Schematic image of stress distribution on abutment screw in Model 2, with load applied at an angle of 90° at 6mm from the center of the implant

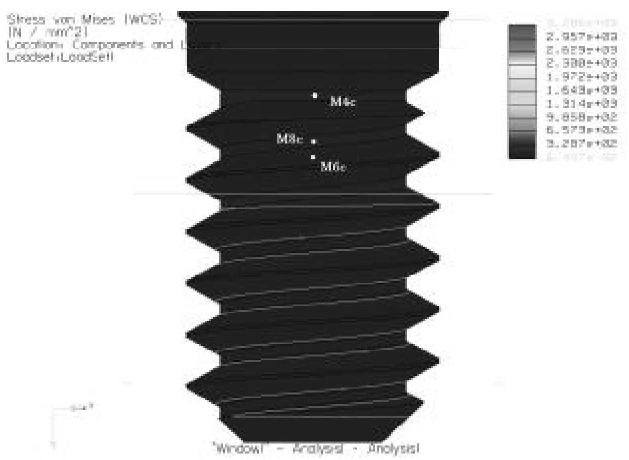

Fig. 6. Schematic image of stress distribution on abutment screw in Model 2, with load applied at an angle of 90° at 6mm from the center of the implant

internal prosthetic connection [7, 28], which may prolong the useful life of the abutment screws, however, it could prejudice the integrity of the implant in the long term, and could precipitate fracture by fatigue.

CONCLUSIONS

When analyzing the stress concentration values of internal connection implants in different abutment screws, it was possible to conclude that:

- The type of material the abutment screw is made of did not have influence on the stress distribution along the prosthetic connection.
- It was not possible to identify the most harmful load situation for the studied points, as the behavior of the stress concentrations did not suggest a pattern.
- The simulations made suggest that the internal connection protects the abutment screw from the accumulated stresses; however, it exposes the implant walls to these stresses.

Analysis by finite elements was shown to be a versatile and promising methodology for analyzing stress concentrations in implant dentistry, but it is worth emphasizing that the FEA (Finite Element Analysis) is an approximate virtual simulation of clinical situations, presenting certain limitations.

REFERENCES

1. Carvalho W, Casado PL, Caśla AL, Barboza EP. Implants for Single First Molar Replacement: Important Treatment Concerns. *Implant Dent* 2006;13:328-35.
2. Jemt T, Laney WR, Harris D, Henry PJ, Krogh PHJ, Polizzi G, et al. Osseointegrated Implants for Single Tooth Replacement: A 1-year Report from a Multicenter Prospective study. *Int J Oral Maxillofac Implants* 1991;6:29-36.
3. Cibirka RM, Steven KN, Brien L, Rueggeberg FA. Examination of the implant – abutment interface after fatigue testing. *J Prosthet Dent* 2001;85:268-75.
4. Norton RM. In Vitro Evaluation of the strength of the conical Implant-to-abutment joint in two commercially available implant systems. *J Prosthet Dent* 2000;83:567-71.
5. Ëehreli MC, Akēa K, Iplikēioglu H, Sahin S. Dynamic fatigue resistance of implant abutment junction in an internally notched morse-taper oral implant: influence of abutment design. *Clin Oral Implant Res* 2004;15:459-65.
6. Lang LA, Kang B, Wang RF, Lang BR. Finite element analysis to determine implant preload. *J Prosthet Dent* 2003;90:539–45.
7. Binon P. Implants and Components: Entering the New Millennium. *Int J Oral Maxillofac Implants* 2000;15:77-93.
8. Geng JP, Tan KBC, Liu GR. Application of finite element analysis in implant dentistry: A review of literature. *J Prosthet Dent* 2001; 85:585-607.
9. Helkimo E, Carlsson GE, Helkimo M. Bite force and state of dentition. *Acta Odontol Scand* 1977;35:297-303.
10. Geramy A, Morgano SM. Finite element analysis of three designs of an implant-supported molar crown. *J Prosthet Dent* 2004;92:434-40.
11. O'Mahony A, Bowles Q, Woolsey G, Robinson SJ, Spencer P. Stress Distribution in the Single unit Osseointegrated Dental Implant: Finite Element analyses of Axial and Off Axial Loading. *Implant Dent* 2000;9:207-18.
12. Binon P, Sutter F, Beaty K, Brunski J, Gulbransem H, Weiner R, et al. The Role of Screw in Implant Systems. *Int J Oral Maxillofac Implants* 1994;9(suppl):48-63.
13. Watanabe F, Uno I, Hata Y, Neuendorff G. Analysis of stress Distribution in a Screw-Retained Implant Prosthesis. *Int J Oral Maxillofac Implants* 2000;15:209-18.
14. Wiskott HWA, Pavone AF, Scherrer SS, Renevey RR, Belser UC. Resistance of ITI Implant Connectors to Multivectorial Fatigue Load Application. *Int J Prosthodont* 2004;17:672-9.
15. Goodacre CJ, Bernal G, Rungcharassaeng K, Kan JYK. Clinical complications with implants and implants prostheses. *J Prosthet Dent* 2003;90:121-32.
16. Weinstein AM, Klawitter J, Anand SC, Schuessle RR. Stress Analysis of Porous Rooted Dental Implants. *J Dent Res* 1976;55:772-7.
17. Satoh T, Maeda Y, Komiyama Y. Biomechanical Rationale for intentionally Inclined Implants in the Posterior Mandible Using 3D Finite Element Analysis. *Int J Oral Maxillofac Implants* 2005;20:533-39.
18. Tada S, Stegaroiu R, Kitamura E, Miyakawa O, Kusakari H. Influence of Implant Design and Bone Quality on Stress/Strain Distribution in Bone Around Implants: A 3-dimensional Finite Element Analysis. *Int J Oral Maxillofac Implants* 2003; 18:357-68.
19. Stegaroiu R, Kusakari H, Nishiyama S, Miyakawa O. Influence of Prosthesis Material on Stress Distribution in Bone and Implant: A 3-Dimensional Finite Element Analysis. *Int J Oral Maxillofac Implants* 1998;13:781-90.
20. Barbier, L; Vander Sloten, J; Krzesinski, G; Schepers, E; Van Der Perre, G. Finite element Analysis of Non–axial Loading versus Axial loading of Oral Implants in Mandible of dog. *J Oral Rehabil* 1998;25:847-58.
21. Himmlovį L, Dostįlovį T, Kacóvskż A, Konvičkovį S. Influence of Implant Length and Diameter on stress distribution: A Finite Element Analysis. *J Prosthet Dent* 2004;91:20-5.
22. Bozkaya D, Muftu S, Muftu A. Evaluation of load transfer characteristics of five implants in compact bone at different load levels by finite element analysis. *J Prosthet Dent* 2004;92:523-30.
23. 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.
24. Akour SN, Fayyad MA, Nayfeh JF. Finite element analyses of two antirotational designs of implant fixtures. *Implant Dent* 2005;14:77-80.
25. Alkan I, Sertgöz A, Ekici B. Influence of occlusal forces on stress distribution in preloaded dental implant screws. *J Prosthet Dent* 2004;91:319-25.
26. Versluis A, Korioth TWP, Cardoso AC. Numerical Analysis of a Dental Implant System Preloaded with a Washer. *Int J Oral Maxillofac Implants* 1999;14:337-41.
27. Martin WC, Woody RD, Miller BH, Miller AW. Implant abutment screw rotations and preloads for four different screw material and surfaces. *J Prosthet Dent* 2001;86:24–32.
28. Khraisat A, Stegaroiu R, Nomura S, Miyakawa O. Fatigue resistance of two implant/abutment joint designs. *J Prosthet Dent* 2002;88:604-10.
29. Boggan RS, Strong JT, Misch CE, Bidez MW. Influence of hex geometry and prosthetic table width on static and fatigue strength of dental implants. *J Prosthet Dent* 1999;82:436-40.
30. Jörnéus L, Jemt T, Carlsson L. Loads and Designs of Screw Joints for Single Crowns Supported by Osseointegrated Implants. *Int J Oral Maxillofac Implants* 1992;7:353-9.

Received: 28 03 2007
Accepted for publishing: 19 05 2009