Influence of Cusp Inclination on Stress Distribution in Implant-Supported Prostheses. A Three-Dimensional Finite Element Analysis

Rosse Mary Falcón-Antenucci, DDS, MSc,[1] Eduardo Piza Pellizzer, DDS, MS, PhD,[1] Paulo Sergio Perri de Carvalho,[2] Marcelo Coelho Goiato, DDS, MS, PhD,[1] & Pedro Yoshito Noritomi, MSc, PhD[3]

[1] Department of Dental Materials and Prosthodontics, Sao Paulo State University, UNESP, Aracatuba, Brazil
[2] Department of Surgery, Sao Paulo State University, UNESP, Aracatuba, Brazil
[3] Renato Archer Information Technology Center, Campinas, Brazil

Keywords
Finite element method; dental implant.

Correspondence
Rosse Mary Falcón-Antenucci, Department of Dental Materials and Prosthodontics, São Paulo State University, UNESP, Dental Materials and Prosthodontics, Rua José Bonifácio 1193, Vila Mendoça, Araçatuba, São Paulo 16015-050, Brazil. E-mail: rosse_falcon@yahoo.com.br, rossefalcon@gmail.com

Accepted June 2, 2009

doi: 10.1111/j.1532-849X.2010.00582.x

Abstract

Purpose: The aim of this study was to assess the influence of cusp inclination on stress distribution in implant-supported prostheses by 3D finite element method.
Materials and Methods: Three-dimensional models were created to simulate a mandibular bone section with an implant (3.75 mm diameter × 10 mm length) and crown by means of a 3D scanner and 3D CAD software. A screw-retained single crown was simulated using three cusp inclinations (10°, 20°, 30°). The 3D models (model 10d, model 20d, and model 30d) were transferred to the finite element program NeiNastran 9.0 to generate a mesh and perform the stress analysis. An oblique load of 200 N was applied on the internal vestibular face of the metal ceramic crown.
Results: The results were visualized by means of von Mises stress maps. Maximum stress concentration was located at the point of application. The implant showed higher stress values in model 30d (160.68 MPa). Cortical bone showed higher stress values in model 10d (28.23 MPa).
Conclusion: Stresses on the implant and implant/abutment interface increased with increasing cusp inclination, and stresses on the cortical bone decreased with increasing cusp inclination.

Dental implants are frequently used in the treatment of edentulous patients, increasing the treatment possibilities in oral rehabilitation. The high success rate and the great number of patients treated with osseointegrated dental implants over the past 20 years have attracted the interest of clinicians and researchers worldwide.[1] Despite the excellent long-term results afforded by the use of implants in dental treatment, they are not free of mechanical complications.[2-4]

The biomechanical aspects of dental implants are quite different from those of natural teeth, due to the capacity of the periodontal ligament[5,6] to absorb stress and permit tooth micromovement, compared to the osseointegrated implant, which has none. There is a possibility that overloads transferred to the implant and surrounding bone could exceed physiologic limits and jeopardize the health of the implant as well as the supported prosthesis.[2-4] Therefore, it is necessary to optimize the distribution of occlusal loads between the prosthesis, the implant, and the surrounding bone.[7,8] Load transfer at the bone/implant interface depends on the type of loading,[9,10] the material properties of the implant and prosthesis, the quality and quantity of the surrounding bone,[10] the implant geometry (length, diameter, and shape),[11-15] and the implant surface structure.[11]

Some analyses have pointed out that the occlusal configuration and cusp inclination of implant-supported prostheses play a significant role in force transmission and the stress-strain relationship between the prosthesis and the bone.[8,13,16] Cusp inclination could increase lateral forces when vertical loads are applied on occlusal surfaces.[7,13,17] Therefore, it is important to not only consider both axial (vertical load) and horizontal forces during stress analysis of the dental implants, but also to consider the more realistic case of combined loads (oblique load) since for a given force these will cause the highest localized stress in cortical bone.[18,19]

The analysis of mechanical behavior in response to stress can be accomplished using techniques such as photoelasticity, strain gauge measurements, and finite element analysis (FEA).[12] When the evaluation involves complex geometries,

Table 1 Material properties

Materials	Young's modulus (E) (GPa)	Poisson's ratio (ν)	References
Trabecular bone	1.37	0.30	Farah et al[21]
Cortical bone	13.7	0.30	Farah et al[21]
Titanium	110.0	0.35	van Rossen et al[22]
NiCr alloy	206.0	0.33	Anusavice & Hojjatie[23]
Feldsphatic porcelain	82.8	0.35	Peyton & Craig[24]

the numerical procedures of FEA provide a more detailed interpretation of the mechanical behavior.[12,18]

Although numerous papers have described the biomechanics of implant-supported prostheses, studies of the influence of occlusal configuration on load transmission at the implant/bone interface are limited and realized with methodology that does not provide detailed stress distributions and the evaluation of specific factors. Therefore, the aim of this study was to assess the influence of cusp inclination on stress distribution in implant-supported prostheses by 3D finite element method.

Materials and methods

Three 3D models were created to simulate a mandibular bone section with an implant and crown. A bone block, 25.46 mm tall, 13.81 mm wide, and 13.25 mm thick, was obtained by computerized tomography of a sagittal section of the second molar region. This bone block was used to represent a section of trabecular bone surrounded by 1 mm of cortical bone. The series of 2D tomographic slices was transferred to InVesalius software (CTI, Campinas, São Paulo, Brazil) for conversion into a 3D model. The resulting 3D model was transferred to 3D CAD software (Rhinoceros 3.0 NURBS Modeling for Windows, Robert McNeel & Associates, Seattle, WA) for further processing. The implant geometry (3.75-mm diameter × 10.00-mm length) was parameterized in a similar fashion from specifications for a screw-shaped dental implant (Conexão Sistemas de Prótese, Arujá, Brazil), and was designed using 3D CAD software (SolidWorks Corp, Concord, MA).

A screw-retained single crown was simulated using three cusp inclinations (10°, 20 °, 30°). The crown framework was constructed of nickel-chromium alloy. Feldspathic porcelain (2-mm thick) was used for the occlusal surface.[9] The crown design, modeled after an artificial second mandibular molar from a dental mannequin, was digitized using a 3D scanner (MDX-20, Roland DG, São Paulo, Brazil). The images were exported to Rhinoceros 3.0 3D CAD software for modeling, and occlusal surface details were added using SolidWorks 3D CAD software. Molar measurements were based on the anatomy of a natural tooth.[20]

The 3D models (models 10d, 20d, and 30d) were transferred to finite element software (NEiNastran 9.0, Noran Engineering, Inc., Westminster, CA). Structural properties, such as Young's modulus and Poisson's ratio, were obtained for each material from the literature[21-24] (Table 1) and incorporated into the model structures. All materials were presumed to be linearly elastic, homogeneous, and isotropic.

The finite element program generated the mesh of the models using a tetrahedral parabolic solid element. The meshes contained approximately 175,000 elements and 285,000 nodes. Boundary conditions were established by fixing the bone block in the X, Y, and Z directions by the lateral faces, leaving the base freely suspended.

The oblique load (45°) of 200 N was determined from the literature[25,26] as the mean value recorded in patients with endosseous implants. A wide range of vertical loads and forces have been reported for patients with endosseous implants, with mean values ranging from 91 N to 284 N. The loads appear to be related to the implant location and food consistency,[25,26] indicating that the limits of the acceptable loads applied to implants are difficult to estimate. The load application was realized such that the cusp inclination did not change the load angle.

Analyses were performed using NEiNastran 9.0 finite element software running on a workstation (Sun Microsystems Inc., São Paulo, Brazil) containing an AMD 64-bit dual core Opteron processor, 4 GB RAM, and a 250 GB hard drive. The results were visualized using von Mises tension maps to display stress values and patterns of stress concentration.

Results

General maps

The maximum stress areas were located at the point of load application. Cross-sections revealed stress propagation from the interface between the crown and the retaining screw to the first or second thread of the implant (Fig 1). Enlarged images of the implant/crown area displayed high-stress concentrations in the framework (NiCr) at the implant platform/crown/retaining screw interface (Fig 2). Stress concentrations increased in proportion to cusp inclination in all models. The von Mises stresses ranged from 26.25 MPa to 56.88 MPa in model 10d, 34.38 MPa to 78.13 MPa in model 20d, and 40.63 MPa to 100 MPa in model 30d.

Cortical bone

Stresses were concentrated at the level of the second to third implant screw thread in the proximal implant contour and the cortical bone surrounding the implant (Fig 3). In all models, enlarged views of the cortical bone region revealed that the maximum stress was located within the cortical bone surrounding the implant at the distal mandibular contour (Fig 4). These maximum stress values were 28.23 MPa in model 10d, 27.98 MPa in model 20d, and 22.51 MPa in model 30d. An increase in inclination from 10° to 20° decreased the stress by only 1%; however, when the inclination was increased to 30°, the stress decreased by 20% (Table 2).

Trabecular bone

The patterns of stress concentration were similar for all models and showed lower stress values when stress was concentrated at the implant apex (Fig 5).

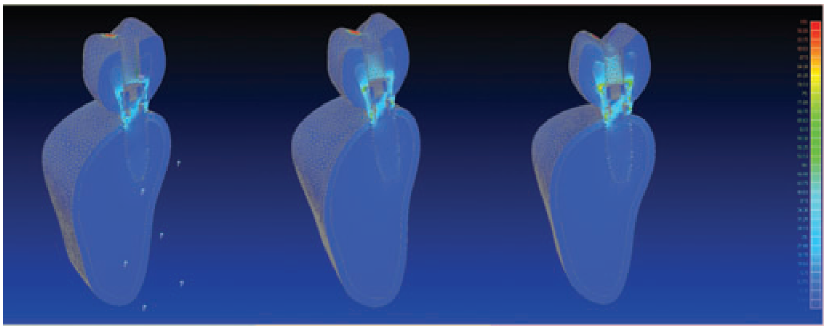

Figure 1 Cross-section of general maps (models 10°, 20°, 30°).

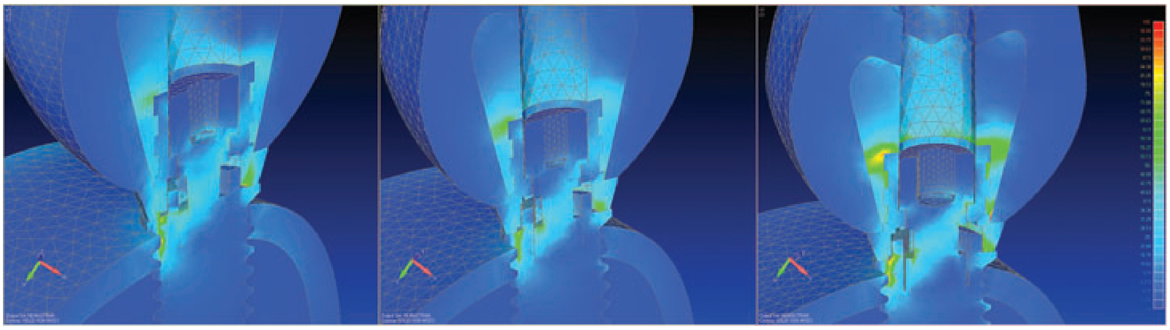

Figure 2 Zoom interface crown/implant (models 10°, 20°, 30°).

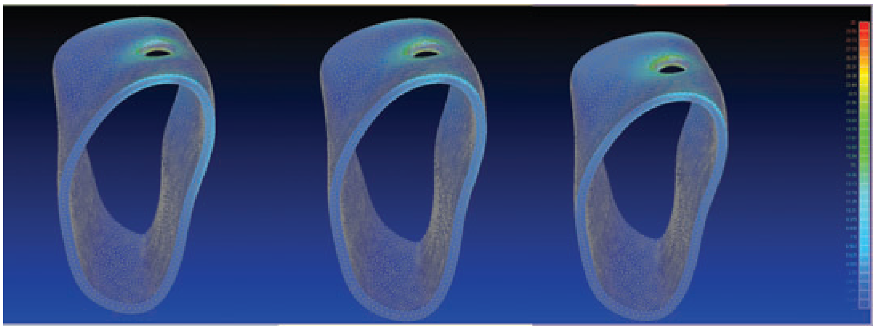

Figure 3 Cortical bone (models 10°, 20°, 30°).

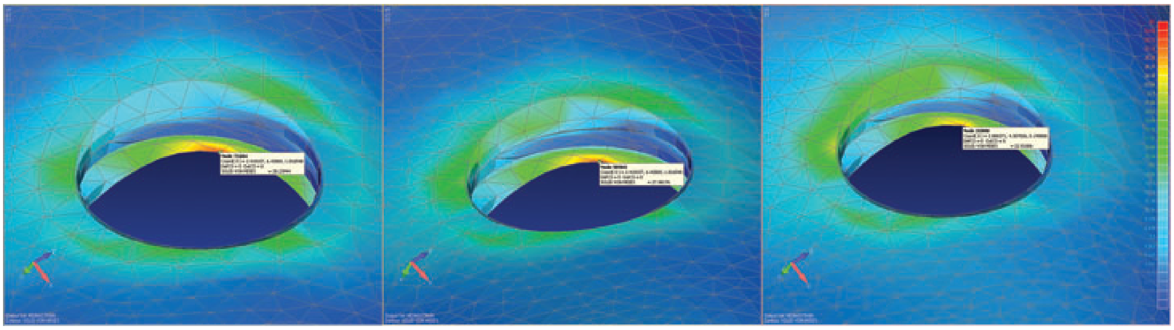

Figure 4 Zoom cortical bone (models 10°, 20°, 30°).

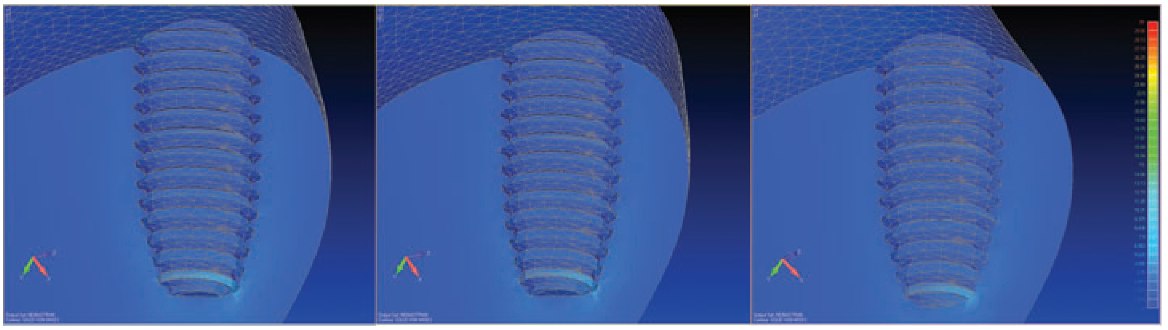

Figure 5 Trabecular bone (models 10°, 20°, 30°).

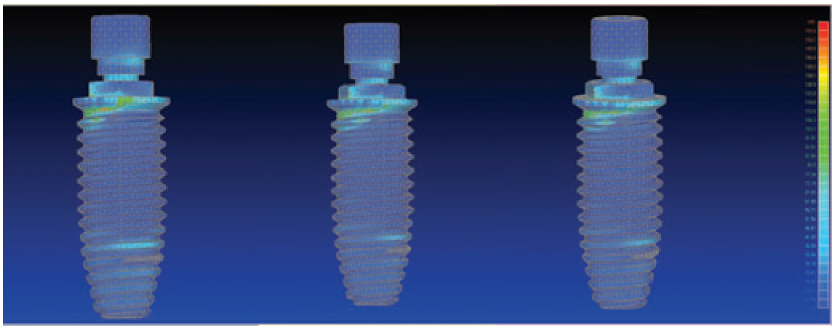

Figure 6 Implant (models 10°, 20°, 30°).

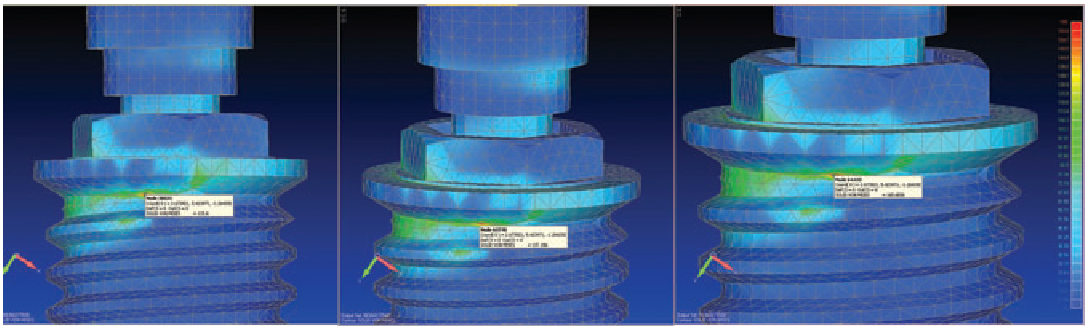

Figure 7 Zoom, implant (models 10°, 20°, 30°).

Implant

Figure 6 depicts the stress concentrated at the implant neck and the abutment. Maximum stress values were found between the implant platform and first screw thread in all models, with 160.68 MPa for model 30d, 137.18 MPa for model 20d, and 115.6 MPa for model 10d. In all models (Fig 7), the stress was concentrated around the retaining screw neck, abutment, and platform implant (range 34.38 MPa to 46.88 MPa in model 10d, 48.13 MPa to 87.5 MPa in model 20d, and 56.72 MPa to 108.3 MPa in model 30d), as well as around the taper of the implant (range 15.63 MPa to 31.25 MPa in model 10d, 13.13 MPa to 30.63 MPa in model 20d, and 20.63 MPa to 36.09 MPa in model 30d). For every 10° of cusp inclination, the stress increased by approximately18% (Table 2).

Table 2 Maximum stress values (MPa) in the models analyzed

Model	10 d	20 d	30 d
Cortical bone	28.23	27.98	22.51
Implant	115.6	137.18	160.68

Discussion

In the analysis of the general map, high-stress concentrations were observed at the implant/crown interface. These stresses were transferred through the screw to the first or second thread of the implant, similar locations to where screw loosening and fracture have been reported by clinical studies[2-4,7] in which mechanical complications were frequently located at the abutment screw, particularly in single crowns.

Maximum stress areas at the crown/implant interface were located in the metal framework (NiCr alloy), probably because the Young's modulus of NiCr[23] is higher than titanium[22] and thus can better tolerate loads. The stress in the crown/implant interface increased with increasing cusp inclination, with the highest stresses appearing in model 30d. According to previous studies, this result is strongly influenced by occlusal contact, and cusp inclination increases the stress on the implant and related components.[8,13,17]

Stress concentrations in the cortical bone occurred mainly at the neck of the implant, corroborating the results of other finite element studies evaluating the stress in implants modeled with and without a supporting framework.[9,10,15,18,27] Cortical bone has a higher modulus of elasticity than trabecular bone and is stronger and more resistant to deformation. For this reason, cortical bone will bear more load than trabecular bone in clinical situations[27] and create a rigid connection with the implant.[8,9] Retrospective clinical studies[2,4] have described cortical bone loss as the result of high-stress concentrations due to overloading. Other authors have attributed bone loss to the generation of a biologic width adjacent to the implant.[28,29] According Abrahamsson et al[28] a certain width of the periimplant mucosa is required to enable a proper epithelial-connective tissue attachment, and if this soft tissue dimension is not satisfied, bone remodeling will progress until the biologic width has been created and stabilized.

The findings from this study suggest that the 30° cusp inclination appears to offer the most effective biomechanical solution, to reduce cortical bone loss due to excessive stress. When the cusp inclination is larger, the resultant line of force falls further away from the rotation center of the implant,[5,8] producing higher stress in the implant and decreasing stress in the cortical bone as the implant absorbs the occlusal loads. Diminished deformation energy in a structure often increases the loading in other structures.[14] In this case, the stress is absorbed by the implant, reducing stress concentrations on other structures such as the cortical bone.

The results obtained for trabecular bone revealed low-stress concentrations. Similar results have been described in other studies in which trabecular bone displayed low-stress concentrations at the implant apex,[15,27] most likely because the low elastic modulus of trabecular bone allows it to absorb transferred loads. Some authors have reported that while the maximum stress concentrations in cortical bone are located in the area of contact with the implant, the maximum stress concentrations in trabecular bone occur around the apex of the implant.[6,9]

The highest stress concentration in the implant occurred at the neck, between the platform of the implant and the first thread. Other studies with FE and mathematical analyses also noted that lateral or inclined forces applied to an implant produce maximum stress at the neck of the implant or at the level of the third screw and crestal bone.[8-10,15] A small stress concentration area was also observed next to the tapered portion of the implant. Previous studies have demonstrated that larger stresses are placed on the supporting bone with tapered implants than with cylindrical implants.[19]

In the current study, an increase in cusp inclination from 10° to 20° produced an increase in stress of 17%, and an increase of 20° to 30° produced an increase in stress of 19%, or an average of 18% increase in bending moment is observed for every 10° increase in cusp inclination. Weinberg and Kruger[13] mathematically argued that cusp inclination is the most potent factor in producing bending moments, and that for every 10° increase in cusp inclination, there was approximately a 30% increase in loading to the implant/prosthesis. The difference between these values could be explained by the difference in analytical methods (mathematical and 3D FEA) and the fact that mathematical analysis represents the internal faces of the cusps as mathematical planes, a simplification that could significantly modify the results.

The use of 3D FEA in the present study allows representation of a more detailed and complex geometry, even though finite element models have their limitations because the mechanical properties and the nonlinear behavior of biological tissues cannot be accurately predicted. On the other hand, finite element models have the advantage of allowing the evaluation of specific factors without the influence of other variables and provide detailed stress distributions.

Some investigators have suggested that the maintenance and preservation of osseointegration are aided by limitation of lateral force transmission by means of reduction of cusp inclination;[7,8] however, the present study demonstrated that a cusp inclination of 30° could be used in a single implant-supported prostheses, if permitted by the occlusal scheme of the patient, because stresses on the cortical bone decreased with increasing cusp inclination. This finding suggests that reducing cusp inclination to decrease occlusal contact and avoid overloading the implant could be unnecessary. Higher stress values in the implant do not jeopardize osseointegration because the increased stress is still lower than the endurance limit of commercial titanium implants (259.90 MPa). This was verified by Clelland et al[15] in a finite element study in which the maximum stresses applied to the implant were near the elastic limit.

Many dental professionals reduce cusp inclination to decrease occlusal contact and avoid overloading the implant; however, this could diminish masticatory efficiency. The findings of Kaukinen et al[17] and Khamis et al[16] indicated that masticatory efficiency improves with increasing cusp inclination. The load on the implants could be influenced by occlusal contact and cusp inclination of the restoration. If contact is allowed during excursive motions of the jaw, the cusp inclination determines the relation between axial forces and lateral forces.[30] The chewing stroke has a lateral component exerted on teeth through the food bolus, whether the teeth actually contact or not. When the bolus can no longer escape by deformation, it exerts similar lateral force to the teeth as if the teeth were together. A parafunction such as bruxism or heavy clenching may introduce a substantial increase in force level, as well as number of loading cycles; however, during bruxism, a slight canine rise will eliminate lateral torque on posterior implants and is advisable. The exception would be when poor alveolar bone supports a natural canine, or when a canine location is an implant site.[5]

Further studies should be undertaken using different methodologies with the aim of elucidating the influence of cusp inclination in implant-supported prostheses, in order to improve masticatory efficiency.

Conclusion

The results of this FEA suggest the following:

1. Stresses on the implant and implant/abutment interface increased with increasing cusp inclination.
2. Stresses on the cortical bone decreased with increasing cusp inclination.

Acknowledgments

The authors thank Conexão Sistemas de Prótese Ltda. for the valuable help received in this research.

References

1. Brånemark PI: Osseointegration and its experimental background. J Prosthet Dent 1983;50:399-410
2. Ekfeldt A, Carlsson GE, Börjesson G: Clinical evaluation of single-tooth restorations supported by osseointegrated implants: a retrospective study. Int J Oral Maxillofac Implants 1994;9:179-183
3. Goodacre CJ: Clinical complications with implants and implant prostheses. J Prosthet Dent 2003;90:121-132
4. Rangert B, Krough HJ, Langer B, et al: Bending overload and implant fracture: a retrospective clinical analysis. Int J Oral Maxillofacial Implants 1995;10:326-334
5. Weinberg LA: The biomechanics of force distribution in implant-supported prostheses. Int J Oral Maxillofac Implants 1993;8:19-31
6. Skalak R: Biomechanical considerations in osseointegrated prostheses. J Prosthet Dent 1983;49:843-848
7. Rangert BO, Jemt T, Jorneus L: Force and moments on Branemark implants. Int J Oral Maxillofac Implants 1989;4:241-247
8. Weinberg LA: Reduction of implant loading with therapeutic biomechanics. Implant Dent 1998;7:277-285
9. Eskitascioglu G, Usumez A, Sevimay M, et al: 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;91:144-150
10. Papavasiliou G, Kamposiora P, Bayne SC, et al: 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
11. 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
12. Şahin S, Çehreli MC, Yalçin E: The influence of functional forces on the biomechanics of implant-supported prostheses-a review. J Dent 2002;30:271-282
13. Weinberg LA, Kruger B: A comparison of implant prosthesis loading with four clinical variables. Int J Prosthodont 1995;8:421-433
14. Duyck J, Naert LE, Van Oosterwyck H, et al: Biomechanics of oral implants: a review of the literature. Technol Health Care 1997;5:253-273
15. Clelland NL, Ismail YH, Zaki HS, et al: Three-dimensional finite element stress analysis in and around the screw-vent implant. Int J Oral Maxillofac Implants 1991;6:391-398
16. Khamis MM, Zaki HS, Rudy TE: A comparison of the effect of different occlusal forms in mandibular implant overdentures. J Prosthet Dent 1998;79:422-429
17. Kaukinen J, Edge MJ, Lang BR: The influence of occlusal design on simulated masticatory forces transferred to implant-retained prostheses and supporting bone. J Prosthet Dent 1996;76:50-55
18. Holmgren EP, Seckinger RJ, Kilgren LM, et al: 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
19. Petrie CS, Willians JL: Comparative evaluation of implant designs: influence of diameter, length, and taper on strains in the alveolar crest. A three-dimensional finite-element analysis. Clin Oral Impl Res 2005;16:486-494
20. Ash MM: Wheeler's Atlas of Tooth form (ed 5). Philadelphia, PA, Saunders, 1984
21. Farah JW, Craig RG, Meroueh KA: Finite element analysis of a mandibular model. J Oral Rehabil 1988;15:615-624
22. van Rossen IP, Braak LH, de Putter C, et al: Stress absorbing elements in dental implants. J Prosthet Dent 1990;64:198-205
23. Anusavice KJ, Hojjatie B: Stress distribution in metal ceramic crowns with a facial porcelain margin. J Dent Res 1987;66:1493-1498
24. Peyton FA, Craig RG: Current evaluation of plastic in crown and bridge prosthesis. J Prosthet Dent 1963;13:743-753
25. Morneburg TR, Proschel PA: Measurement of masticatory forces and implant forces and implant loads: a methodologic clinical study. Int J Prosthodont 2002;15:20-27
26. Morneburg TR, Proschel PA: In vivo forces on implants influenced by occlusal scheme and food consistency. Int J Prosthodont 2002;16:481-486
27. Papavasilou G, Kamposiora P, Bayne SC, et al: 3D-FEA of osseointegration percentages and patterns on implant-bone interfacial stresses. J Dent 1997;25:485-491
28. Abrahamsson I, Berglundh T, Wennström J, et al: The peri-implant hard and soft tissues at different implant systems. A comparative study in the dog. Clin Oral Implants Res 1996;7:212-219
29. Herman JS, Buser D, Schenk RK, et al: Crestal bone changes around titanium implants. A histomorphometric evaluation of unloaded non-submerged and submerged implants in the canine mandible. J Periodontol 2000;71:1412-1424
30. Balshi TJ, Hernandez RE, Pryszlak MC, et al: A Comparative study of one implant versus two replacing a single molar. Int J Oral Maxillofac Implants 1997;11:372-378

