Acta Odontologica Scandinavica, 2012; 70: 61–71

informa
healthcare

ORIGINAL ARTICLE

Biomechanical comparison of two different collar structured implants supporting 3-unit fixed partial denture: A 3-D FEM study

GÖKÇE MERIÇ[1], ERKAN ERKMEN[2], AHMET KURT[3], ATILIM ESER[4] & AHMET UTKU ÖZDEN[4]

[1]*Near East University, Faculty of Dentistry, Department of Prosthodontics, Mersin 10, Turkey,* [2]*Gazi University, Faculty of Dentistry, Department of Oral and Maxillofacial Surgery Ankara, Turkey,* [3]*Atilim University, Faculty of Engineering, Department of Manufacturing Engineering, Ankara, Turkey, and* [4]*Aachen University, Institute for Materials Applications in Mechanical Engineering, Aachen, Germany*

Abstract

Objective. The purpose of the study was to compare the effects of two distinct collar geometries of implants on stress distribution in the bone as well as in the fixture-abutment complex, in the framework and in the veneering material of 3-unit fixed partial denture (FPD). ***Material and methods.*** The 3-dimensional finite element analysis method was selected to evaluate the stress distribution in the system composed of 3-unit FPD supported by two different dental implant systems with two distinct collar geometries; microthread collar structure (MCS) and non-microthread collar structure (NMCS). In separate load cases, 300 N vertical, 150 N oblique and 60 N horizontal, forces were utilized to simulate the multidirectional chewing forces. Tensile and compressive stress values in the cortical and cancellous bone and von Mises stresses in the fixture-abutment complex, in the framework and veneering material, were simulated as a body and investigated separately. ***Results.*** In the cortical bone lower stress values were found in the MCS model, when compared with NMCS. In the cancellous bone, lower stress values were observed in the NMCS model when compared with MCS. In the implant-abutment complex, highest von Mises stress values were noted in the NMCS model; however, in the framework and veneering material, highest stress values were calculated in MCS model. ***Conclusions.*** MCS implants when compared with NMCS implants supporting 3-unit FPDs decrease the stress values in the cortical bone and implant-abutment complex. The results of the present study will be evaluated as a base for our ongoing FEA studies focused on stress distribution around the microthread and non-microthread collar geometries with various prosthesis design.

Key Words: *dental stress analysis, implant, prosthodontics*

Introduction

In recent years, as a result of advances in oral implantology, osseointegrated dental implants have been shown to be predictable options for treatments ranging from the replacement of single tooth to complete arch restorations [1]. Threaded, root-form osseointegrated implants were developed by Brånemark in the late 1960s and were initially designed to support complete arch, fixed implant-supported restorations for completely edentulous patients [2]. However, in the last decade, these types of implants have also been successfully used to support fixed partial dentures (FPD).

Dental implants must fulfill certain criteria arising from special demands of function, which include biocompatibility, adequate mechanical strength, and transmission of functional forces to bone, within physiological limits [3].

In all incidences of functional loading with implants, the occlusal forces are transferred to the bone–implant interface via implant supported prosthesis. The process and the consequences of force transmission into supporting bone depends on the nature of applied force, the design of implants (micro and macro architecture), the biological and biomechanical properties of the bone–implant interface, and the reaction of bone tissue to the mechanical environment created by loading the

Correspondence: Gökçe Meriç, DDS, PhD, Near East University, Faculty of Dentistry, Department of Prosthetic Dentistry, Mersin 10, Turkey.
Tel: +90 533 8426443. Fax: +90 392 6802025. E-mail: gokcemeric@yahoo.com

(Received 30 April 2010; revised 16 February 2011; accepted 24 March 2011)
ISSN 0001-6357 print/ISSN 1502-3850 online
DOI: 10.3109/00016357.2011.597775

implant [4]. One of the most critical elements influencing the long-term uncompromised functioning of an oral implant is its own design [5].

As confirmed by several clinical studies, osseointegrated implants can fail mainly as a consequence of bone weakening or loss of bone in the peri-implant region [6]. Bone resorption process affects mainly the implant neck region and can be activated by surgical trauma or bacterial infection as well as overloading of the bone–implant interface [7]. Several previous studies have attempted to minimize crestal bone resorption by increasing the contact area of bone to the implant interface and, therefore, reducing stress at the cortical alveolar crest [8,9]. Thus, attempts to increase the contact area of the bone–implant interface have focused on the shape and characteristics of the implant surface or altering the fixture design or shape at the collar region [8,9]. Thread form and configuration are important objectives in biomechanical optimization. Threads are used to maximize initial contact, improve initial stability and to enlarge the implant surface area and a favorable dissipation of the interfacial stress [10]. Thread depths, thickness, pitches and helix angle are some of the geometric variables that determine the functional thread surface and affect the biomechanical load distribution of the implant [10]. For this reason, several implant concepts have been developed and many implant types are commercially available. Moreover, the retention element, for example in the form of a thread at the endosseous neck portion of the implant would induce the mechanical stimulation required to maintain the marginal bone level [11].

The highest bone stresses have been reported to be concentrated in the cortical bone in the region of the implant neck [8]. The transfer of functional loads and accompanying stress distribution in a bone–implant–denture assembly depends on the physical properties and special geometry of each component and therefore stress analysis of bone–implant mechanical interactions is important [12]. Considering not only the effectiveness and reliability of endosseous implants, but also the expanded indications for implants and changing clinical protocols, the relation between implant design and load distribution at the implant–bone interface are important issues [9].

Biomechanical aspects of implant-supported FPDs are currently under investigation and a number of *in vitro*, clinical and animal studies have attempted to predict the clinical behavior of dental applications associated with implant supported dentures [13–15]. *In vitro* methods include photoelastic analysis, conventional *in vitro* model analysis, and finite element model analysis (FEA).

FEA has been used extensively in medicine and dentistry in the past few decades [16]. This method of analysis has proved valuable by improving the profession's understanding of various aspects of oral biomechanics [16]. Studies comparing the accuracy of these analyses found that, if detailed stress information is required either for absolute or relative comparison, then 3D modeling will be necessary [17]. The 3D FEA is considered an appropriate method for investigation of the stress throughout a three-dimensional structure and it was selected for bone stress evaluation in the current study.

The hypothesis of the study was based on a systematic analysis of the effect of different collar geometries upon the bone stresses under conventional 3-unit FPDs.

Thus the aim of this study was to evaluate the stress transfer properties of two currently available implants that differ significantly in macroscopic collar geometry which support 3-unit fixed partial prosthesis. This approach allowed the evaluation of load transfer characteristics under vertical, horizontal and obliquely loaded prosthesis by using 3-dimensional finite element analyses.

Material and methods

A 3-dimensional model of a totally edentulous mandibular bone was used as the basis of a mandibular finite element model in this study. The modeled section of the mandible was composed as a dense cancellous core surrounded by a thick layer of cortical bone. The average thickness of the cortical bone in the crestal area was 2.0 mm. Serial axial sections in every 0.5 mm level of an edentulous mandible was obtained from NewTom 3G (Quantitative Radiology, Verona, Italy) Cone-Beam CT (CBCT) imaging System. The CBCT images were stored using DICOM 3.0 as a medical image file format and imported into Maxilim Software (Medicim Company, Mechelen, Belgium) version 2.2.2, a 3D medical image processing programme. The 3D image of the mandible with the (.stl) file format was imported into MSC Mentat (MSC Software Corporation, CA, USA) version 2005 for pre-processing and modeling.

In the absence of information regarding the precise organic material properties of bone and the other materials modeled and used in this study they were considered to be isotropic, homogeneous, and linearly elastic. As long as a relative comparison is made between the FEA results, an assumption will yield to accurate comments on the subject. The elastic properties were taken from the literature and shown in Table I [18,19].

Instead of the isotropic and homogeneous material modeling, different Young's modulus values can be calculated from the density values in Hounsfield units and implemented to the FE model. Studies show that [3,5] Hounsfield units are good enough to describe the tendency of Youngs's modulus compared with the change in density but fail to calculate the absolute value of it with accuracy. There is, for some values,

Table I. Mechanical properties of materials used in the study.

Material	Young modulus (GPa)	Poisson ratio (v)
Cortical bone	14.8	0.30
Cancellous bone	1.85	0.30
Ti-6Al-4V	110	0.32
Cobalt–chromium alloy	220	0.30
Feldisphatic porcelain	61.2	0.19

more than 100% difference between Young's modulus for the same density. So it is concluded that it will be dangerous to compare and comment on the absolute values of stresses and strains found in FE analysis. One of the critical inputs to the analysis, material characterization, may change drastically even for a single element used in the computation.

A preliminary verification study has been done to support the assumptions and facts described above. The results showed that using homogeneous or heterogeneous material properties over a body will change the "absolute" values but will not change the "relative" manner of the resultant stresses. If a model generates more stresses than the other, it is independent of the homogenity or heterogenity of the material properties. Since the object of the current study is "to compare" different geometries in a "relative" manner, homogeneous material properties, which are assumed to represent the overall and average material behaviour, have been assigned for either cortical or cancellous along the entire bone segment.

The cement film thickness which does not affect the retention capabilities of cements is represented to change between 25–100 μm [20]. Such a film thickness is not expected to have an effect on the overall result.

Implants were embedded in the same location on the selected mandibular segment for both models (Figure 1a). The implants were rigidly anchored along their entire interface and bonded in the bone to mimic good osseointegration and to inhibit any relative movement between the bodies, i.e. the implants were perfectly embedded into the structure and fulfilled their aim of usage perfectly. Similarly, a type of rigid bond was provided at the implant–abutment and abutment–prosthesis interfaces to simulate the real fix connection conditions between these components (Figure 1b).

In the current study, a model of a 14.0 mm long and 4.1 mm in diameter solid–screw *bone-level* ITI implant (Straumann AG, Waldenburg, Switzerland) for non-microthread collar structure (NMCS) and a model of a 13.0 mm long and 4.0 mm in diameter solid–screw Astra Tech implant for the microthread collar structure (MCS) (Astra-Tech AB, Mölndal, Sweden) were used (Figure 2). Additionally, a cementable abutment RC for ITI and cementable direct abutments for the Astra implants were selected. The geometry of the implants as well as the abutments were modeled according to engineering drawings by using MSC Mentat. Therefore, in both implant systems the main thread was constituted as a single continuous spiral. In the microthread implant system the microthread portion of the implant was constructed with two additional spirals, running with a 120° angle to each other as identified by the manufacturer. Thus, in the microthread portion of the implant one pitch was divided in to three pitches equally. Cobalt–Chromium (Bego, Bremen, Germany) was used as the framework material and feldspathic porcelain was used for the veneering material. The thickness of the metal framework and porcelain used in this study were 0.5 mm and 1.5 mm, respectively, and the cement thickness was ignored. All the final solid meshes were constituted by tetrahedral elements with four nodes by using Msc Marc (MSC. Software Corporation, CA, USA).

The present study was performed by using 3D FEA. As there were no appropriate experimental results in the literature to compare our results to validate the finite element model, six different models for each implant

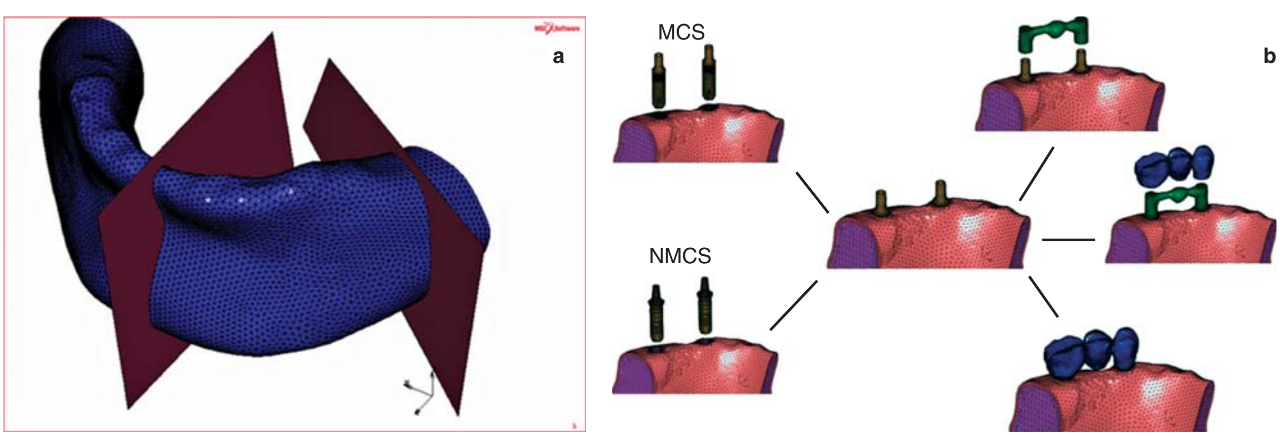

Figure 1. (a) The selected mandibular segment for both models where the implants embedded on. (c) 3D finite element models of mandibular bone, implant–abutment complex, framework design and fixed partial denture.

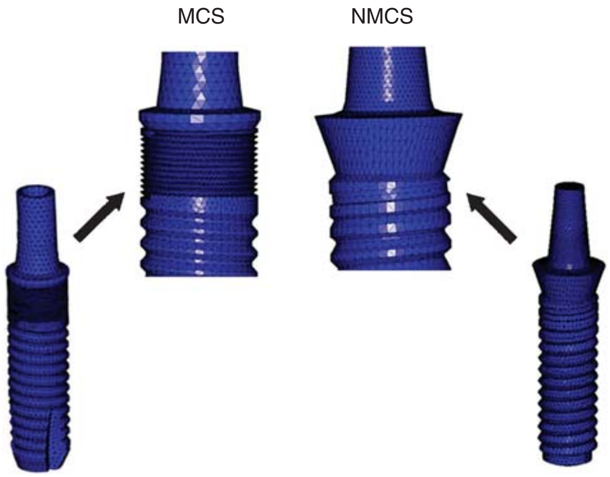

Figure 2. 3D finite element model of MCS (implant with microthread collar structure) and NMCS (implant with non-microthread collar structure).

system with changing number of elements were compared and the convergence of the results were examined using the result of the maximum tensile stress in cortical bone under vertical load condition (Figure 3). The results and the number of the elements are shown in Table II. From the table the convergence for the two models were reached for the following element numbers and the result of those models were considered in the rest of the study. Thus, our 3D models consist of 230,117 elements and 44,199 nodes for the NMCS model and 217,514 elements and 42,148 nodes for the MCS model.

A ratio of 5:2.5:1 was found for vertical (V), oblique (O) and horizontal (H) loads, respectively, during chewing by Graf et al. [21]. Therefore, 300 N static vertical loads were applied along the central fossa of the crowns. To simulate the oblique load condition, an oblique static load of 150 N was applied to the buccal cusps of the crowns with an inclination of 60° buccally from the vertical; 60 N static loads were applied in a mesiodistal direction horizontally to mimic the parafunctional movement of the jaw. The models were separately analyzed in a MSC Marc (MSC Software Corporation, CA, USA) finite element solver for the vertical, oblique and horizontal load sets. The stress contours were computed and plotted in the cortical and cancellous bone as well as in the fixture–abutment complex, in the framework and in the veneering material.

Results

In the study, von Mises stress was utilized on implants, metal framework and veneering material. Maximum (tensile stress) and minimum (compressive stress) principle stress (P_{max} and P_{min}) on bony structures were used to assess the stress distribution in the modeled system. The von Mises stress values and P_{max} and P_{min} values are shown in Figures 4–6.

A color scale with 12 stress values served to measure quantitatively the stress distribution in the implant–abutment complex, framework and veneering material as well as the cortical and cancellous bone. The scaling was selected not to represent the yield strength but rather to provide clear visualization of the region of stress.

Fixture–abutment complex

The highest stress value has been noted in NMCS models under horizontal loads and localized at the collar of the fixtures (Figure 4). In MCS models, the highest von Mises stress values have been determined very close to each other in the case of all three loading conditions and the stress values were found to be less than the NMCS models (Figure 4). Under the all-loading conditions, the stresses have been distributed along the microthreads rather than the fixture–abutment junction in the MCS model. However, in the NMCS model; stresses were noted

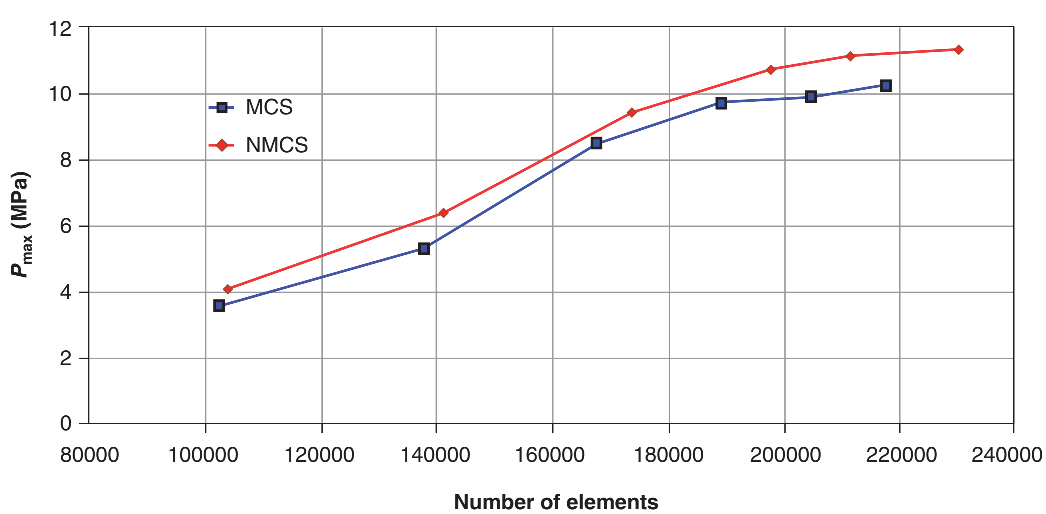

Figure 3. Converge of the two models with the changing numbers of elements and nodes.

Table II. Principal maximum stress values for vertical load on cortical bone with changing number of elements for convergence and final numbers of elements and nodes.

# of elements for MCS model	102,347	137,816	167,518	189,216	204,628	217,515
# of elements for NMCS model	103,918	141,227	173,721	197,724	211,514	230,117
	Principal maximum stress (P_{max}) (maximum tensile stress) MPa					
Microthread vertical load case	3.6	5.3	8.5	9.7	9.9	10.2
Non-microthread vertical load case	4.1	6.4	9.4	10.1	11.1	11.3
	MCS model			NMCS model		
Final # of elements	217,514			230,117		
Final # of nodes	42,148			44,199		

around the fixture–abutment junction and on the collar threads (Figure 7). The stress distribution patterns were similar on first and second implant–abutment complex in both MCS and NMCS models (Figures 7a, 7b).

Framework

Comparison of the NMCS and MCS models from the standpoint of frameworks has revealed that higher von Mises stresses were found in the MCS models within the vertical, horizontal and oblique loads respectively (Figure 4). The highest stress values were observed in the MCS model under the vertical load. Dispersion of the stress in the frameworks in both models seemed very similar (Figure 8a, 8b). Under the vertical load, the mesial frame showed higher stress on the lingual surface while lesser stress concentrates mesiolingually on the distal one in the MCS model (Figure 8a). Stress was also concentrated at the mesial and distal connectors under the vertical load. Within the horizontally loaded models, stress was mainly concentrated around the collar area of both mesial and distal frames and the connectors while, within the obliquely loaded models, stress was only concentrated around the collar of the mesial and distal frames.

Veneering material

The highest stress value was found in MCS model around the collar region of the premolar crown spreading on the lingual surface in case of vertical loading. In the MCS models, the highest von Mises stress values have been noted in the vertical load case and followed by horizontal and oblique load cases respectively (Figure 4). The most affected part among the whole model was premolars' veneering structure in the MCS models. Stresses were concentrated in the coronal third of the premolar's lingual aspect in MCS model (Figure 9a). Similarly, in the NMCS model stresses were concentrated both at the coronal and middle third of the premolar's lingual side (Figure 9b). Under the oblique load, stress was observed at the collar and connector regions of the mesial and distal veneering parts in both models.

Cortical bone

In the cortical bone, the highest P_{max} stresses values have been noted in the NMCS model in the case of oblique load condition towards distally and disto-lingually around the distal implant neck rather than the mesial one (Figure 5). Under the horizontal load, stress was concentrated at the mesiolingual aspect of

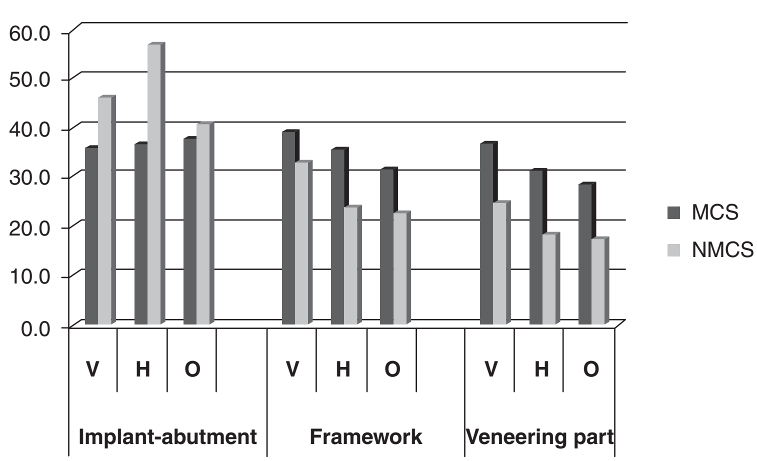

Figure 4. Highest von Mises stress values recorded at the implant–abutment, framework and veneering part.

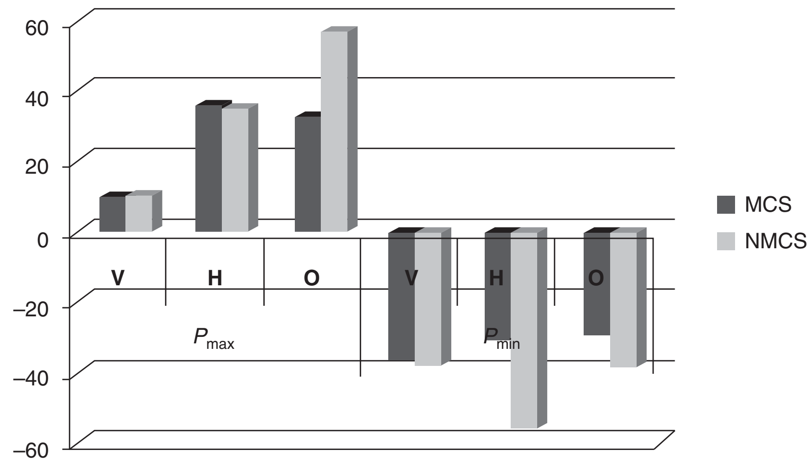

Figure 5. Principal maximum and principal minimum (tensile maximum and compression maximum) values recorded at the cortical bone.

mesial implant, and the distolingual aspect of distal implant. Stress concentration patterns have been found to be similar to each other in MCS and NMCS models in all load conditions (Figures 10a, 10b). Under such similar load distribution patterns, the implant geometry also affects the absolute value of the stress which is localized under the same points. The comparison of NMCS and MCS models in respect of P_{min} stress values revealed that in all three loading conditions the highest values have been recorded in the NMCS models (Figure 5). The distal and mesial housings surrounded by cortical bone were affected similarly in aspect of stress distribution patterns in both MCS and NMCS models under the applied loads (Figures 10C and D).

Cancellous bone

In the cancellous bone, either the P_{max} or the P_{min} stress values in the NMCS models have been identified less than in the MCS models in all loading conditions (Figure 6). Additionally, the stress patterns have been traced similarly to each other in both models (Figures 11a, b). The highest P_{max} stresses were found in the MCS model at the lingual sides of both implants within the horizontal load case (Figure 11A). In the vertical loading condition the P_{max} stress has been concentrated at the collar area extending all around the mesial and distal housings in the direction of distally and distobuccally to affect the distal one slightly more in both models. The highest P_{max} stress was located on the lingual aspect of the cancellous bone near microthread flanges under oblique loads. According to the values that have been calculated, NMCS and MCS models showed that the highest P_{min} stress values were noted in the MCS model within the horizontal load case. The P_{min} values in the case of horizontal loads was concentrated at the buccal sides of both implants (Figures 11c, d).

Discussion

In recent years in order to improve osseointegration, studies have focused on implant material, shape, and surface characteristics [22]. However, the role of bioengineering in dentistry cannot be overlooked.

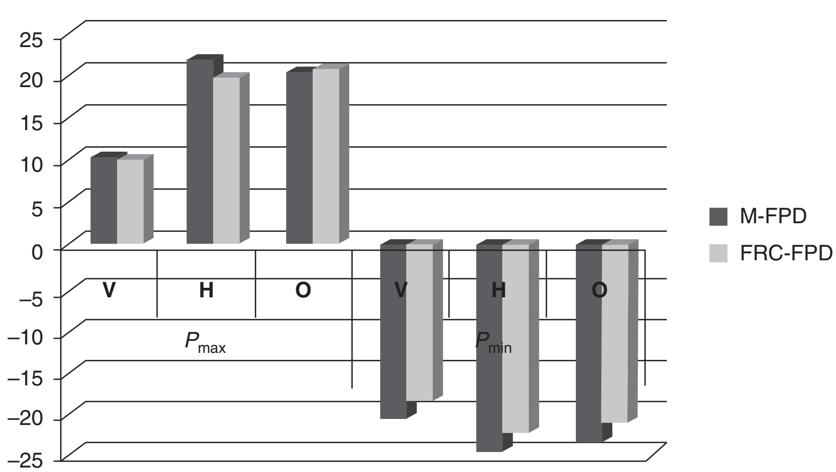

Figure 6. Principal maximum and principal minimum (tensile maximum and compression maximum) values recorded at the cancellous bone.

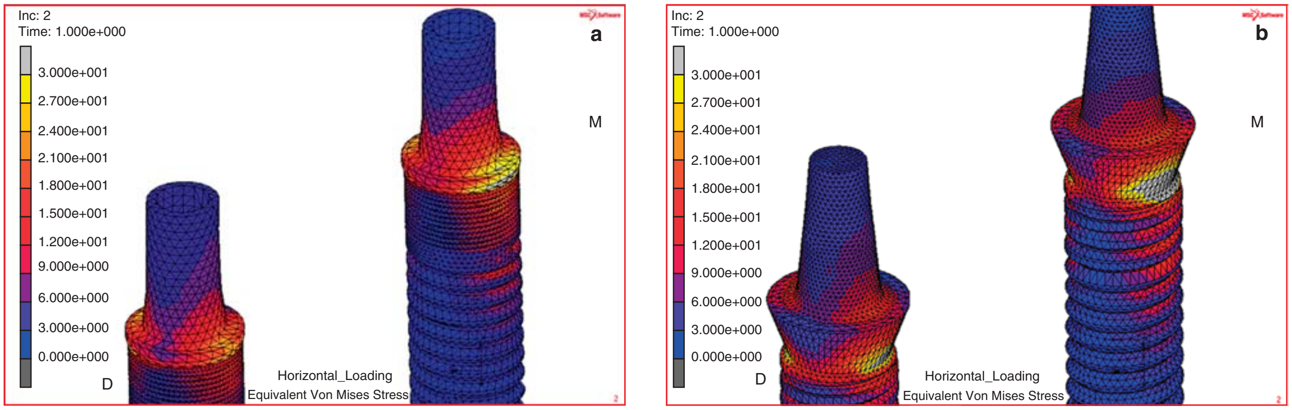

Figure 7. von Mises stress values and stress distribution patterns recorded in the fixture–abutment complex under horizontal load case. (a) MCS model; (b) NMCS model.

Bioengineering studies have been useful in understanding the response of biologic tissues and bone to force [23]. There are an increasing number of studies targeting the cause of failure in clinical practice, using various stress analysis methods [7–9]. In the present study a 3-dimensional FEA was used to evaluate the stress transfer properties of two currently-available implants that differ with their collar geometry individually which support conventional 3-unit FPDs. The aim of this study was not to replicate exact *in vivo* stresses but rather to illustrate a possible difference between microthread and non-microthread collar structures.

The use of FEA in implant dentistry has been widely demonstrated and published. Due to the geometries involved with modeling implants and alveolar process are very complex, FEA is considered the most suitable tool for analyzing them. Despite, some FEA studies' results have been achieved from axisymmetric models and the authors believed that the different models used and the combination of the different modifications introduced in the modified axisymmetric models satisfactorily mimics a broad range of mandibular anatomies occurring in a clinical practice. A comparison of 2-D and 3-D FEA in a study showed that only 3-D FEA can realistically simulate the stress pattern in space [24]. Thus, in the current study a 3-D finite element model was used to compare the stress distribution in the posterior mandibular segment around the implants which had distinct collar geometries.

When applying FEA to dental implants, it is important to consider not only axial loads and horizontal forces (moment-causing loads) but also a combined load (oblique occlusal force) because the latter represents more realistic occlusal directions and, for a given force, will result in localized stress in the cortical bone [25]. Therefore, in the present study vertical, horizontal, and oblique static loads were applied to the superstructures.

The mechanical behavior of bone is difficult to model as it is highly anisotropic, has heterogeneous characteristics and is dependent on many parameters such as age, sex, type of bone [26]. Consequently, it is not easy to introduce the correct material properties of the specific bone that is numerically studied. That is the reason why, in most finite element models, the mechanical properties of bone are supposed to be isotropic [27]. In the present study, several assumptions and simplifications have been made with regard

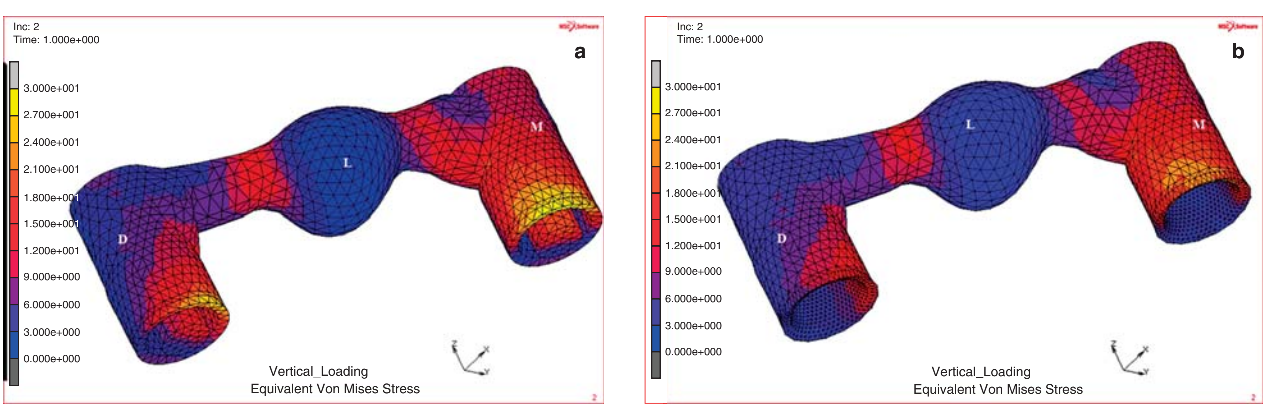

Figure 8. von Mises stress values and stress distribution patterns recorded in the framework under vertical load. (a) MCS model; (b) NMCS model.

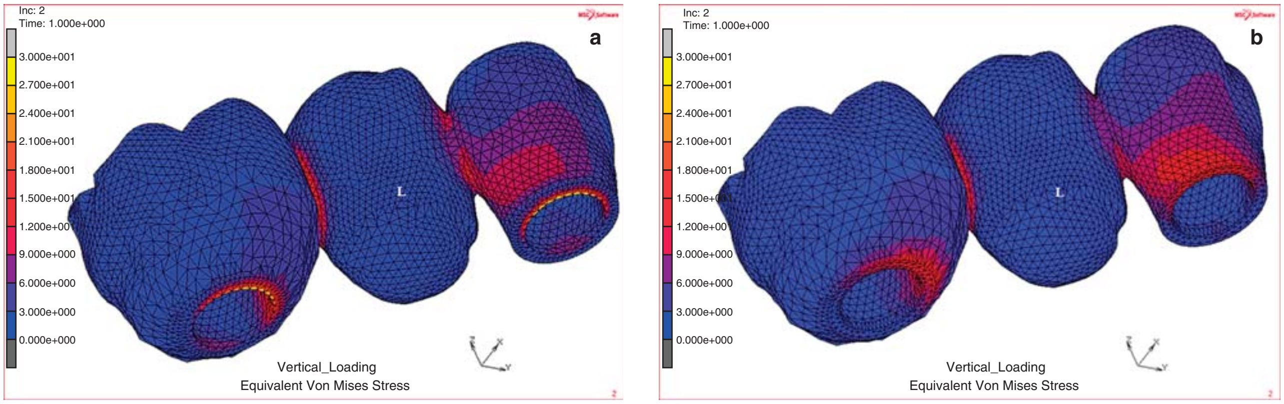

Figure 9. von Mises stress values and stress distribution patterns recorded in the veneering material under vertical load. (a) MCS model; (b) NMCS model.

to the material properties and model generation. The properties of the materials modeled in the study, particularly the living tissues, however, are different. For example, it is known that the actual cortical bone of the mandible is transversally isotropic and inhomogeneous [12]. Additionally, in a real clinical situation, the mandible is individually unique and presents a great many differences in respect of shape and geometry as well as bone qualities and consequently material properties such as Young modulus and Poisson ratio in its different portions. An estimation of the absolute stresses value occurring in a real case requires the use of the actual anatomy of the patient itself. Nevertheless, in the present study a relative biomechanical evaluation has been aimed at comparing the effects of different collar-shaped implants at the same selected mandibular location. Thus, these variables would not affect the study result and these parameters can be ignored. Moreover, the structures in the model were all assumed to be homogenous, isotropic and linearly elastic and implants were assumed to be 100% osseointegrated. Furthermore, some ultrastructural clinical and FEM studies indicated that there is never a 100%

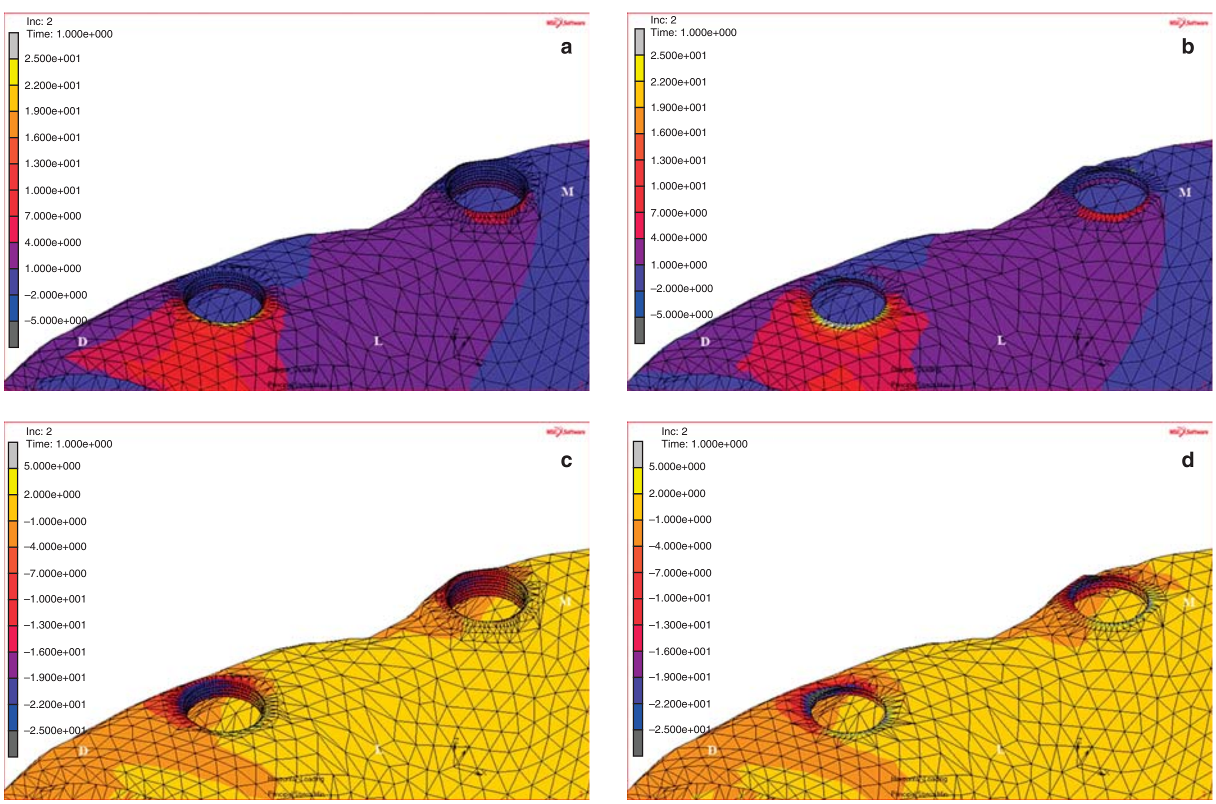

Figure 10. P_{max} and P_{min} stress values and stress distribution patterns recorded in the cortical bone. (a) P_{max} for MCS model under oblique load; (b) P_{max} for NMCS model under oblique load; (c) P_{min} for MCS model under horizontal load; (d) P_{min} for NMCS model under horizontal load.

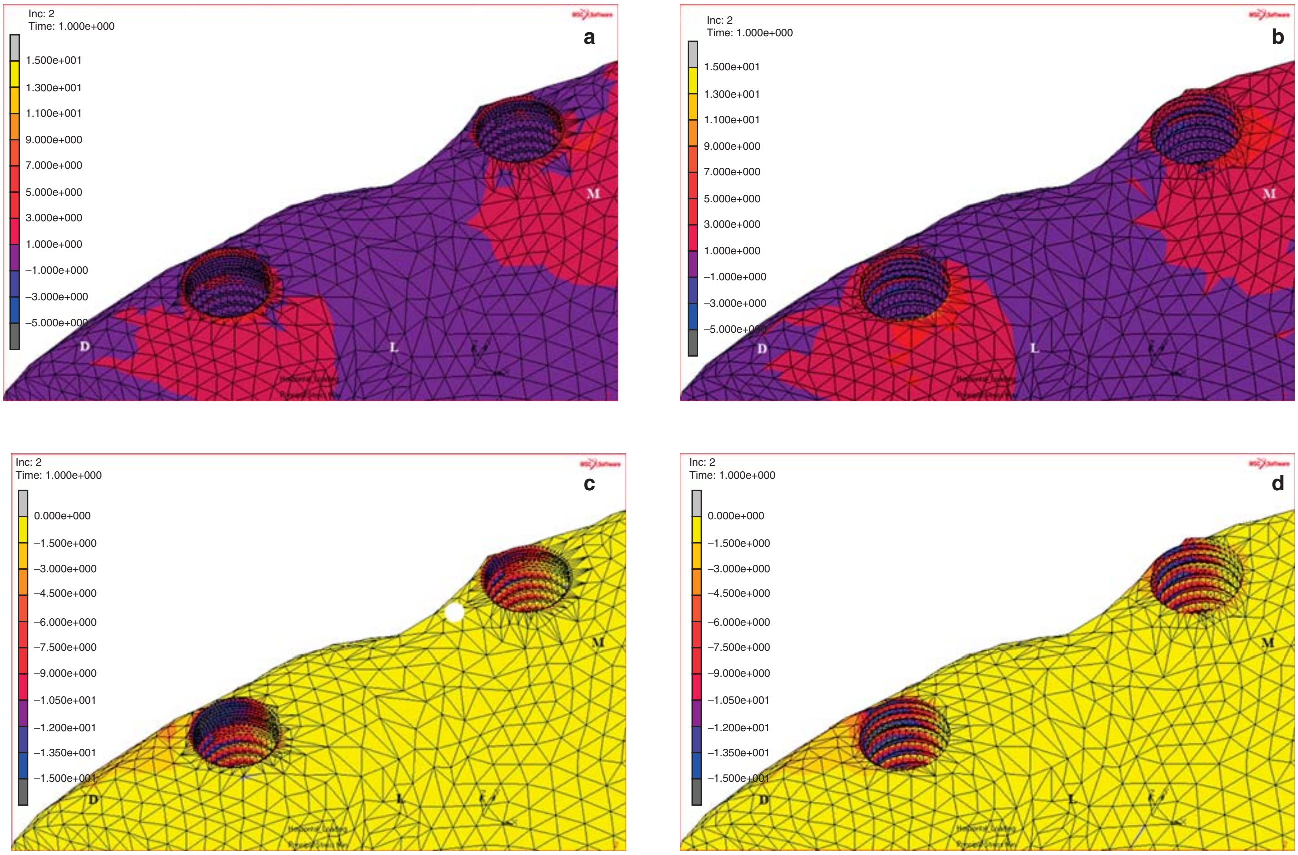

Figure 11. P_{max} and P_{min} stress values and stress distribution patterns recorded in the cancellous bone. (a) P_{max} for MCS model under horizontal load; (b) P_{max} for NMCS model under horizontal load; (c) P_{min} for MCS model under horizontal load; (d) P_{min} for NMCS model under horizontal load.

bone–implant interface [12]. Therefore, inherent limitations of the FEA must be acknowledged.

The results of the FEA to some extent depend upon the size of the elements. If the purpose is to obtain accurate values for the stresses, the element mesh should be refined at locations with large stress gradients. However, the purpose of this study was to compare the stresses at the collar geometry for different implant designs. In such a situation it is sufficient if the element mesh is identical at the different thread turns for the different designs [28]. The models used in the current study fulfilled these requirements.

In the present study, the boundary fixations included constraining all three degrees of freedom at each of the nodes located at the most external mesial or distal aspects of the model. Such a fixation is necessary to enable a statical problem to be solved; otherwise the problem will be a dynamical one which does not represent the actual case. As the boundary conditions (loading and fixation) are far away from the implant–bone interface where we had the special interest, referring to Saint Venant's principle (the localized effect of any loading will smooth out within regions that are sufficiently away from the location of the load) and the works of Sato et al. [29].

The design of the implant–abutment interface affects the stress distribution in the marginal bone. In the present study, it was concluded that when the implant–abutment junction is close to the crestal bone level, it will increase the magnitude of stresses and strains at that region, which in turn may cause bone loss. Accordingly, internal cone designs have been shown to provide lower bone–implant interfacial stresses rather than butt–joint designs using FEA [30]. Therefore, either NMCS or MCS examples with their specific internal design of morse taper with 8° for NMCS and conical with 11° for MCS interface configurations were utilized in our study to evaluate the stress distribution.

The implant collar is one of the most important parts that is directly related to biomechanical problems such as biological failures or fractures. In successful osseointegrated implants, the initial breakdown of the implant–tissue interface generally begins at the crestal region. Crestal bone loss has been documented as an important factor that affects the long-term longevity of a dental implant. Our recorded principal stress values for MCS, except P_{max} under the horizontal load, are lower than the stress values for NMCS at the cortical bone. It is well known that if bone is subjected to extreme stresses it is resorbed [31]. The retentive form at the coronal portion of the fixture has favorably reduced the progressive marginal bone stresses by the occlusal load at the margin. The ultimate tensile and compressive strengths of cortical bone have been reported as 133 and 193 MPa, respectively [32]. Maximum tensile and compressive stresses in this study in the cortical bone were lower than these values, which revealed no area to be critically loaded with respect to mechanical stresses.

On the other hand, according to Wolff's law, the stresses within the physiological tolerance range act as a stimulus for new bone formation. Hansson [33] suggested that retention elements at the implant neck, as opposed to a smooth neck, may counteract marginal bone resorption. Palmer et al. [34] reported no significant bone loss in their observations of the implants using a microthreaded collar design. Similar findings were presented by Karlsson et al. [35] and Lee et al. [36].

In the present study, it was evaluated that stresses tended to be concentrated at the cortical bone around the neck of the implant closest to the load, whereas stresses in cancellous bone were considered low. This is probably due to the fact that the elastic modulus of cortical bone is higher than cancellous bone and that cortical bone is more resistant to deformation.

The stresses on implants, abutments, framework, and veneering structure are also important to evaluate because these structures, being the stiffest components of an implant prosthodontic system, bear a great amount of stress and are responsible for transmitting the load to the bone. A well-planned and well-executed prosthesis is essential to avoid excessive and unnecessary forces on bone and implant components. Mechanical integrity of the prosthesis and implant combined to the framework rigidity is essential to the longevity of the prosthesis [37].

However, as the two implant systems shared mostly an identical fixture–abutment junction, a difference in the design of the implant collar mainly would lead to distinctive features at the stress distribution in the bone, at the implant–abutment complex, framework and veneering part. According to the manufacturers, the ITI philosophy designates two implants with a 4.1 mm diameter to support a 3-unit fixed prosthesis from each end for the placement of the missing three posterior teeth [38]. In the current study, a 14-mm long implant is used to demonstrate the NMCS and a 13-mm long implant for the MCS accordingly. The diameters of the modeled implants were 4.1 mm for NMCS and 4.0 mm for the MCS. Nevertheless, some clinical studies demonstrated that the success rate of an implant was proportional to the implant length, but some authors observed that the length of the implant had little influence on the amount of stress levels [39]. In the current study it has been targeted to remain with the original product lines of the manufacturers. The changes in geometry of the objects may alter the biomechanical behavior. Hence, the 1 mm longer NMCS implant used in the current study can cause a leverage effect when compared with the MCS implant. However, this adverse effect may be compensated with the extended diameter in NMCS. Moreover, in such a comparative study, the implant lengths and diameters used in the entire study were assessed as a fixed parameter and could easily be ignored.

In this study, the highest von Mises stresses have been noted in NMCS models in the implant–abutment complex, whereas the MCS models had the highest von Mises stresses at the framework and veneering material. This was due to the difference in load transfer mechanisms depending on the collar design of the implant systems. These stresses at the framework and veneering material are lower than the yield strengths of the materials (cobalt-chromium alloy = 720 MPa, feldspathic porcelain = 500 MPa) [18,19]. They do not seem to have any clinical relevance on the mechanical as well as the biomechanical outcome. The NMCS implant collar may be prone to bend and create mechanical stress within the implant–abutment interface. This result suggests that implant retained prosthesis with a microthreated collar designed fixture, avoiding excessive and unnecessary stresses on the implant–abutment complex.

In conclusion, data obtained in this study showed that the collar geometry of dental implants, which support 3-unit FPDs significantly, affects the load transfer mechanism and stress distribution patterns not only in cortical and cancellous bone but also in the implant abutment complex alone, framework and veneering material. In the current study, it has been noted that the implants with microthread collar structure when compared with the non-microthread collar geometry under 3-unit FPDs decrease the stress values in the cortical bone and the implant–abutment complex. Additionally, a 3-unit FPD supported by two implants was considered as an ideal approach for the restoration of an edentulous jaw portion in a prosthodontic point of view, some questions may arise in minds regarding the stress distribution patterns with the different types of fixed prosthetic designs supported by two microthread and non-microthread collar dental implants. Therefore, the present study results will be evaluated as a base for our ongoing FEM studies focusing on such problems.

Declaration of interest: The authors report no conflicts of interest. The authors alone are responsible for the content and writing of the paper.

References

[1] Christensen GJ. Implant prosthodontics: from single tooth to complex cases. J Oral Implantol 2002;28:244–8.

[2] Zarb GA, Schmitt A. The longitudinal clinical effectiveness of osseointegrated dental implants: the Toronto study. Part III: problems and complications encountered. J Prosthet Dent 1990;64:185–94.

[3] Albrektsson T, Brånemark PI, Hansson HA, Lindström J. Osseointegrated titanium implants. Requirements for ensuring a long-lasting, direct bone-to-implant anchorage in man. Acta Orthop Scand 1981;52:155–70.

[4] Cehreli M, Sahin S, Akça K. Role of mechanical environment and implant design on bone tissue differentiation: current knowledge and future contexts. J Dent 2004;32:123–32.

[5] Geramy A, Morgano SM. Finite element analysis of three designs of an implant-supported molar crown. J Prosthet Dent 2004;92:434–40.

[6] Eckert SE, Wollan PC. Retrospective review of 1170 endosseous implants placed in partially edentulous jaws. J Prosthet Dent 1998;79:415–21.

[7] Brunski JB. Biomechanics of dental implants. In: Block M, Kent JN, Guerra LR, editors. Implants in dentistry. Philadelphia, PA: WB Saunders; 1997. p.63–71.

[8] Meijer HJ, Kuiper JH, Starmans FJ, Bosman F. Stress distribution around dental implants: influence of superstructure, length of implants, and height of mandible. J Prosthet Dent 1992;68:96–102.

[9] 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.

[10] Kong L, Zhao Y, Hu K, Li D, Zhou H, Wu Z, et al. Selection of the implant thread pitch for optimal biomechanical properties: a three-dimensional finite element analysis. Adv Eng Softw 2009;40:474–8.

[11] Hansson S, Werke M. The implant thread as a retention element in cortical bone: the effect of thread size and thread profile: a finite element study. J Biomech 2003;36:1247–58.

[12] Simşek B, Erkmen E, Yilmaz D, Eser A. Effects of different inter-implant distances on the stress distribution around endosseous implants in posterior mandible: a 3D finite element analysis. Med Eng Phys 2006;28:199–213.

[13] Aparicio C, Orozco P. Use of 5-mm–diameter implants: periotest values related to a clinical and radiographic evaluation. Clin Oral Implants Res 1998;9:398–406.

[14] Gratton DG, Aquilino SA, Stanford CM. Micromotion and dynamic fatigue properties of the dental-implant interface. J Prosthet Dent 2001;85:47–52.

[15] Geramy A, Sharafoddin F. Abfraction: 3D analysis by means of the finite element method. Quintessence Int 2003;34:526–33.

[16] Geng JP, Tan KB, Liu GR. Application of finite element analysis in implant dentistry: a review of the literature. J Prosthet Dent 2001;85:585–98.

[17] Stegaroiu R, Sato T, Kusakari H, Miyakawa O. Influence of restoration type on stress distribution in bone around implants: a three-dimensional finite element analysis. Int J Oral Maxillofac Implants 1998;13:82–90.

[18] Iplikçioğlu H, Akça K. Comparative evaluation of the effect of diameter, length and number of implants supporting three-unit fixed partial prostheses on stress distribution in the bone. J Dent 2002;30:41–6.

[19] Kayabaşi O, Yüzbasıoğlu E, Erzincanli F. Static, dynamic and fatigue behaviours of dental implant using finite element method. Adv Eng Softw 2006;37:649–58.

[20] Wolfart M, Wolfart S, Kern M. Retention forces and seating discrepancies of implant-retained castings after cementation. Int J Oral Maxillofac Implants 2006;21:519–25.

[21] Graf H, Graseel H, Aeberhard HJ. A method for the measurement of occlusal forces in three directions. Helv Odont Acta 1974;18:7–11.

[22] Nagasao T, Kobayashi M, Tsuchiya Y, Kaneko T, Nakajima T. Finite element analysis of the stresses around fixtures in various reconstructed mandibular models–part II (effect of horizontal load). J Craniomaxillofac Surg 2003;31: 168–75.

[23] Akca K, Iplikcioglu H. Evaluation of the effects of the residual bone angulation on implant-supported fixed prosthesis in mandibular posterior edentulism. Part II. 3D finite element stress analysis. Implant Dent 2001;10:238–43.

[24] Ismail YH, Pahountis LN, Fleming JF. Comparison of two-dimensional and three-dimensional finite element analysis of a blade implant. Int J Oral Implantol 1987;4:25–31.

[25] Zhang JK, Chen ZQ. The study of effects of changes of the elastic modulus of the materials substitute to human hard tissues on the mechanical state in the implant-bone interface by three-dimensional anisotropic finite element analysis. West China J Stomatol 1998;16:274–8.

[26] Bonnet AS, Postaire M, Lipinski P. Biomechanical study of mandible bone supporting a four-implant retained bridge: finite element analysis of the influence of bone anisotropy and foodstuff position. Med Eng Phys 2009;31:806–15.

[27] Hansson S, Ekestubbe A. Area moments of inertia as a measure of the mandible stiffness of the implant patient. Clin Oral Implants Res 2004;15:450–8.

[28] Hansson S. A conical implant-abutment interface at the level of the marginal bone improves the distribution of stresses in the supporting bone. An axisymmetric finite element analysis. Clin Oral Implants Res 2003;14:286–93.

[29] Sato Y, Teixeira ER, Shindoi N, Wadomoto M, Akagawa Y. Effect of bone length on stress distribution in implant FEA. J Dent Res 1997;76:328.

[30] Hansson S. Implant-abutment interface: biomechanical study of flat top versus conical. Clin Implant Dent Relat Res 2000;2: 33–41.

[31] Frost HM. Skeletal structural adaptations to mechanical usage (SATMU): 4. Mechanical influences on intact fibrous tissues. Anat Rec 1990;226:433–9.

[32] Reilly DT, Burstein AH. The elastic and ultimate properties of compact bone tissue. J Biomech 1975;8:393–405.

[33] Hansson S. The implant neck: smooth or provided with retention elements. A biomechanical approach. Clin Oral Implants Res 1999;10:394–405.

[34] Palmer RM, Smith BJ, Palmer PJ, Floyd PD. A prospective study of Astra single tooth implants. Clin Oral Implants Res 1997;8:173–9.

[35] Karlsson U, Gotfredsen K, Olsson C. A 2-year report on maxillary and mandibular fixed partial dentures supported by Astra Tech dental implants. A comparison of 2 implants with different surface textures. Clin Oral Implants Res 1998;9: 235–42.

[36] Lee DW, Choi YS, Park KH, Kim CS, Moon IS. Effect of microthread on the maintenance of marginal bone level: a 3-year prospective study. Clin Oral Implants Res 2007; 18:465–70.

[37] Skalak R. Biomechanical considerations in osseointegrated prostheses. J Prosthet Dent 1983;49:843–8.

[38] Buser D, Belser UC, Lang NP. The original one-stage dental implant system and its clinical application. Periodontol 1998; 2000:106–18.

[39] Jemt T. Failures and complications in 391 consecutively inserted fixed prostheses supported by Branemark implants in edentulous jaws: a study of treatment from the time of prosthesis placement to the first annual checkup. Int J Oral Maxillofac Implants 1991;6:270–6.