Application of finite element analysis in implant dentistry: A review of the literature

Jian-Ping Geng, BDS, MSD,[a] Keson B. C. Tan, BDS (Hons), MSD,[b] and Gui-Rong Liu, PhD[c]
Faculty of Dentistry and Faculty of Engineering, National University of Singapore, Singapore

Finite element analysis (FEA) has been used extensively to predict the biomechanical performance of various dental implant designs as well as the effect of clinical factors on implant success. By understanding the basic theory, method, application, and limitations of FEA in implant dentistry, the clinician will be better equipped to interpret results of FEA studies and extrapolate these results to clinical situations. This article reviews the current status of FEA applications in implant dentistry and discusses findings from FEA studies in relation to the bone–implant interface, the implant–prosthesis connection, and multiple-implant prostheses. (J Prosthet Dent 2001;85:585-98.)

In the past 2 decades, finite element analysis (FEA) has become an increasingly useful tool for the prediction of the effects of stress on the implant and its surrounding bone. Vertical and transverse loads from mastication induce axial forces and bending moments and result in stress gradients in the implant as well as in the bone. A key factor for the success or failure of a dental implant is the manner in which stresses are transferred to the surrounding bone. Load transfer from implants to surrounding bone depends on the type of loading, the bone–implant interface, the length and diameter of the implants, the shape and characteristics of the implant surface, the prosthesis type, and the quantity and quality of the surrounding bone. FEA allows researchers to predict stress distribution in the contact area of the implants with cortical bone and around the apex of the implants in trabecular bone.

Although the precise mechanisms are not fully understood, it is clear that there is an adaptive remodelling response of the surrounding bone to this stress. Implant features causing excessive high or low stresses may contribute to pathologic bone resorption or bone atrophy. This article reviews the current status of the application of FEA in implant dentistry. Assumptions made in the use of FEA in implant dentistry are described, and findings from FEA studies are discussed in relation to the bone–implant interface, the implant–prosthesis connection, and multiple-implant prostheses.

[a]Research Scholar, Department of Restorative Dentistry, Faculty of Dentistry.
[b]Associate Professor, Department of Restorative Dentistry, Faculty of Dentistry.
[c]Associate Professor, Centre for Advanced Computations in Engineering Science, Faculty of Engineering.

ASSUMPTIONS IN THE USE OF FEA IN THE IMPLANT-BONE BIOMECHANICAL SYSTEM

For problems involving complicated geometries, it is very difficult to achieve an analytical solution. Therefore, the use of numerical methods such as FEA is required. FEA is a technique for obtaining a solution to a complex mechanical problem by dividing the problem domain into a collection of much smaller and simpler domains (elements) in which the field variables can be interpolated with the use of shape functions. An overall approximated solution to the original problem is determined based on variational principles. In other words, FEA is a method whereby, instead of seeking a solution function for the entire domain, one formulates the solution functions for each finite element and combines them properly to obtain the solution to the whole body. Because the components in a dental implant-bone system are extremely complex geometrically, FEA has been viewed as the most suitable tool for analyzing them. A mesh is needed in FEA to divide the whole domain into elements. The process of creating the mesh, elements, their respective nodes, and defining boundary conditions is referred to as "discretization" of the problem domain.

FEA was initially developed in the early 1960s to solve structural problems in the aerospace industry but has since been extended to solve problems in heat transfer, fluid flow, mass transport, and electromagnetics. In 1976, Weinstein et al[1] were the first to use FEA in implant dentistry; subsequently, FEA was applied rapidly in that field. Atmaram and Mohamed[2-4] analyzed the stress distribution in a single-tooth implant to understand the effect of elastic parameters and geometry of the implant, implant length variation, and pseudo-periodontal ligament incorporation. Borchers and Reichart[5] performed a 3-dimensional FEA of an implant at different stages of bone interface development. Cook et al[6] applied FEA to porous rooted dental implants. Meroueh et al[7] performed an FEA for an osseointegrated cylindrical implant. Williams et al[8] carried out an FEA on cantilevered prostheses on dental implants, and Akpinar et al[9] used FEA to simulate the combination of a natural tooth and implant.

The principal difficulty in simulating the mechanical behavior of dental implants is the modelling of human bone tissue and its response to applied mechanical force. Certain assumptions need to be made to make the modelling and solving process possible. The complexity of the mechanical characterization of bone and its interaction with implant systems has forced authors to make major simplifications. Some assumptions influence the accuracy of the FEA results significantly. These include: (1) detailed geometry of the bone and implant to be modelled,[10] (2) material properties,[10] (3) boundary conditions,[10] and (4) the interface between bone and implant.[11]

Geometry

The first step in FEA modelling is to represent the geometry of interest in the computer. In some 2-dimensional FEA studies, the bone was modelled as a simplified rectangular configuration with the implant.[11-13] Some 3-dimensional FEA models treated the mandible as an arch with rectangular section.[14,15] Recently, with the development of digital imaging techniques, more efficient methods are available for the development of anatomically accurate models. These include the application of specialized software for the direct transformation of 2- or 3-dimensional information in image data from computed tomography (CT) or magnetic resonance imaging (MRI) into FEA meshes. The automated inclusion of some material properties from measured bone density values is also possible.[16,17] This allows more precise modelling of the geometry of the bone–implant system. In the foreseeable future, the creation of FEA models for individual patients, based on advanced digital techniques, will become possible and perhaps even commonplace.

Material properties

Material properties greatly influence the stress and strain distribution in a structure. These properties can be modelled in FEA as isotropic, transversely isotropic, orthotropic, and anisotropic. In an isotropic material, the properties are the same in all directions; therefore, only 2 independent material constants exist. An anisotropic material has different properties when measured in different directions. There are many material constants depending on the degree of anisotropy (transversely isotropic, orthotropic).

In most reported studies, the assumption is made that the materials are homogeneous and linear and that they have elastic material behavior characterized by 2 material constants of Young's modulus and Poisson's ratio. Early FEA studies ignored the trabecular bone network simply because the capability to determine the trabecular pattern was not available. Therefore, it was assumed that trabecular bone has a solid pattern inside the inner cortical bone shell. Both bone types were modelled simplistically as linear, homogeneous, and isotropic materials. A range of different material parameters have been recommended for use in previous FEA studies (Table I).[5,18-33]

Several authors[34-37] have pointed out that cortical bone is neither homogeneous nor isotropic (Table II). This nonhomogenous, anisotropic, composite structure of bone possesses different values for ultimate strain and modulus of elasticity when bone is tested in compression compared with in tension. Test conditions also affect the material properties measured. Rieger et al[12] reported that a range of stresses (1.4 to 5.0 MPa) appears to be needed for healthy maintenance of bone. Stresses outside this range have been reported to cause bone resorption.

Boundary conditions

Most FEA studies modelling the mandible set the boundary conditions as fixed. Recently, Zhou et al[38] developed a more realistic 3-dimensional mandibular FEA model from transversely scanned CT image data. The functions of the mastication muscles and the ligamenteous and functional movement of the temporomandibular joints (TMJ) were simulated by means of cable elements and compressive gap elements, respectively. The authors concluded that cable and gap elements can be used to set boundary conditions in their mandibular FEA model, improving the model mimicry and accuracy. Expanding the domain of the model can reduce the effect of inaccurate modelling of the boundary conditions. This, however, is at the expense of computing and modelling time. Teixera et al[39] concluded that in a 3-dimensional mandibular model, modelling the mandible at distances greater that 4.2 mm mesially or distally from the implant did not result in any significant further yield in FEA accuracy. The use of infinite elements can be a good way to model boundary conditions.

Bone–implant interface

Most FEA models assume a state of optimal osseointegration, meaning that cortical and trabecular bone are assumed to be perfectly bonded to the implant. This does not occur so exactly in clinical situations. Therefore, the imperfect contact and its effect on load transfer from implant to supporting bone need to be modelled more carefully. Current FEA programs provide several types of contact algorithms for simulation of contacts. It is therefore now technically feasible to conduct such a simulation. The friction between

Table I. Material parameters used in finite element analysis studies of dental implants

Material	Elastic modulus (Pa)	Poisson's ratio	Author
Enamel	4.14×10^4	0.3	Davy et al[18]
	4.689×10^4	0.30	Wright and Yettram[19]
	8.25×10^4	0.33	Farah et al[20]
	8.4×10^4	0.33	Farah et al[21]
Dentin	1.86×10^4	0.31	Reinhardt et al[22]
	1.8×10^4	0.31	MacGregor et al[23]
Parodontal membrance	171	0.45	Atmaram and Mohammed[24]
	69.8	0.45	Reinhardt et al[22]
	6.9	0.45	Farah et al[21]
Cortical bone	2727	0.30	Rice et al[25]
	1.0×10^4	0.30	Farah et al[21]
	1.34×10^4	0.30	Cook et al[26]
	1.5×10^4	0.30	Cowin[27]
Trabecular bone	150	0.30	Cowin[27]
	250	0.30	MacGregor et al[23]
	790	0.30	Knoell[28]
	1.37×10^3	0.31	Borchers and Reichart[5]
Mucosa	10	0.40	Maeda and Wood[29]
Pure titanium	117×10^3	0.30	Sakaguichi and Borgersen[30]
Ti-6Al-4V	110×10^3	0.33	Colling[31]
Type 3 gold alloy	100×10^3	0.30	Sakaguichi and Borgersen[30]
	80×10^3	0.33	Lewinstein et al[32]
Ag-Pd alloy	95×10^3	0.33	Craig[33]
Co-Cr alloy	218×10^3	0.33	Craig[33]
Porcelain	68.9×10^3	0.28	Lewinstein et al[32]
Resin	2.7×10^3	0.35	Craig[33]
Resin composite	7×10^3	0.2	Craig[33]

contact surfaces can also be modelled with contact algorithms. The friction coefficients, however, have to be determined through experimentation.

Bone is a porous material with complex microstructure. The higher load-bearing capacity of dense cortical bone compared with the more porous trabecular bone is generally recognized. On implant insertion, cortical and/or trabecular bone, starting at the periosteal and endosteal surfaces, gradually form a partial-to-complete encasement of the implant. However, the degree of encasement is dependent on the stresses generated and the location of the implant in the jaw.[37] The anterior mandible is associated with 100% cortical osseointegration; this percentage decreases toward the posterior mandible. The least cortical osseointegration (<25%) is seen in the posterior maxilla. The degree of osseointegration appears to be dependant on bone quality and stresses developed during healing and function. To study the influence of osseointegration in greater detail at the bone trabeculae contact to implant level, Sato et al[40] set up 4 types of stepwise assignment algorithms of elastic modulus according to the bone volume in the cubic cell (Fig. 1). They showed that their 300 μm element size was valid for modelling the bone–implant interface.

Table II. Anistropic properties of cortical bone

	Cortical shell	
Elastic (MPa)	Diaphyseal	Metaphyseal
Longitudinal	17,500	9,650
Transverse	11,500	5,470

Summary

In summary, stress distribution depends on assumptions made in modelling geometry, material properties, boundary conditions, and the bone–implant interface. To obtain more accurate stress predictions, advanced digital imaging techniques can be applied to model the bone geometry more realistically; the anisotropic and nonhomogenous nature of the material must be considered; and boundary conditions must be carefully treated with the use of computational modelling techniques. In addition, modelling of the bone–implant interface should incorporate the actual osseointegration contact area in cortical bone as well as the detailed trabecular bone contact pattern through the use of contact algorithms in FEA.

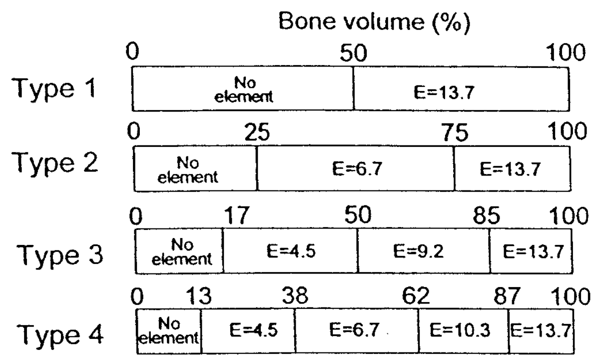

Fig. 1. Four types of stepwise assignment algorithms of elastic modulus according to bone volume in cubic cell. *E* = elastic modulus (GPa). (Reproduced with permission from J Oral Rehabil 1999;26:641.)

THE BONE–IMPLANT INTERFACE

Analyzing force transfer at the bone–implant interface is an essential step in the overall analysis of loading, which determines the success or failure of an implant. It has long been recognized that both implant and bone should be stressed within a certain range for physiologic homeostasis. Overload can cause bone resorption or fatigue failure of the implant, whereas underloading of the bone may lead to disuse atrophy and subsequent bone loss.[41,42] With the use of load cells in rabbit calvaria, Hassler et al[43] showed that the target compressive stress level for maximum bone growth occurs at 1.8 MPa, levelling off to a control level at 2.8 MPa. Skalak[44] stated that close apposition of bone to the titanium implant surface means that under loading, the interface moves as a unit without any relative motion; this is essential for the transmission of stress from the implant to the bone at all parts of the interface.

In centric loading, several FEA studies[45-47] of osseointegrated implants demonstrate that when maximum stress concentration is located in cortical bone, it is in the contact area with the implant, and when the maximum stress concentration is in trabecular bone, it occurs around the apex of the implant. In cortical bone, stress dissipation is restricted to the immediate area surrounding the implant; in trabecular bone, a fairly broader distant stress distribution occurs.

Stress transmission and biomechanical implant design problems

FEA can simulate the interaction phenomena between implants and the surrounding tissues. Analysis of the functional adaptation process is facilitated by the ability to investigate the various loading, implant, and surrounding tissue variables. Load transfer at the bone–implant interface depends on the: (1) type of loading; (2) material properties of the implant and prosthesis; (3) implant geometry, length and diameter as well as shape; (4) implant surface structure; (5) nature of the bone–implant interface; and (6) quality and quantity of the surrounding bone. Most efforts have been directed at optimizing implant geometry to maintain the beneficial stress level in a variety of loading scenarios.

Loading

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 cause the highest localized stress in cortical bone.[48] Barbier et al[49] investigated the influence of axial and nonaxial occlusal loads on the bone remodelling phenomena around IMZ implants in a dog mandible simulated with FEA. A strong correlation between the calculated stress distributions in the surrounding bone tissue and the remodelling phenomena in the comparative animal model was observed. The authors concluded that the highest bone remodelling events coincide with the regions of highest equivalent stress and that the major remodelling differences between axial and nonaxial loading are determined largely by the horizontal stress component of the engendered stresses. The importance of avoiding or minimizing horizontal loads thus was emphasized.

Zhang and Chen[50] compared dynamic with static loading in 3-dimensional FEA models with a range of different elastic moduli for the implant. Their results showed that, compared with the static load models, the dynamic load model resulted in higher maximum stress at the bone–implant interface as well as a greater effect on stress levels when elastic modulus was varied.

In summary, both static and dynamic loading of implants have been modelled with FEA. In static load studies, it is necessary to include oblique occlusal forces to achieve more realistic modelling. Most studies conclude that excessive horizontal force should be avoided. The effects of dynamic loading require further investigation.

Prosthesis and implant material properties

High-rigidity prostheses are recommended because the use of low elastic moduli alloys for the superstructure predicts larger stresses at the bone–implant interface on the loading side than the use of a rigid alloy for a superstructure with the same geometry.[51] Stegariou et al[52] used 3-dimensional FEA to assess stress distribution in bone, implant, and abutment when a gold alloy, porcelain, or resin (acrylic or composite) was used for a 3-unit prosthesis. In almost all situations, stress in the bone–implant interface with the resin prostheses was similar to or higher than that in the models with the other 2 prosthetic materials. However, in his classical mechanical analysis, Skalak[44] stated that the presence of a resilient element in an implant prosthesis superstructure would reduce the high load rates that occur when occluding unexpectedly on a hard object. For this reason, he suggested the use of acrylic resin teeth. Nevertheless, several other studies[53,54] could not demonstrate any significant differences in the force absorption quotient of gold, porcelain, or resin prostheses.

The elasticity moduli of different implant materials also influences the implant–bone interface. Implant materials with too low moduli are to be avoided; Malaith et al[55] suggested that implant materials have a modulus of elasticity of at least 110,000 N/mm^2. Rieger et al[56] indicated that serrated geometry led to high-stress concentrations at the tips of the bony ingrowth and near the neck of the implant. Low moduli of elasticity emphasized these concentrations. The nontapered, screw-type geometry showed high-stress concentrations at the base of the implant when high moduli were modelled and at the neck of the implant when low moduli were modelled. The authors concluded that a tapered endosseous implant with a high elastic modulus would be most suitable for dental implantology. However, the design must not cause high-stress concentrations, which commonly lead to bone resorption, at the implant neck. Stoiber[57] reported that in the construction of an appropriate screw implant, special attention must be paid to the rigidity of the implant rather than to thread design.

In summary, although the effect of prosthesis material properties is still being debated, it is well established that implant material properties greatly affect the location of stress concentrations at the implant–bone interface.

Implant geometry: length, diameter, and shape

Large implant diameters provide for more favorable stress distributions.[55,58] FEA has been used to show that stresses in cortical bone decrease in inverse proportion to an increase in implant diameter with both vertical and lateral loads.[58] However, Holmgren et al[48] showed that using the widest diameter implant is not necessarily the best choice when considering stress distribution to surrounding bone; within certain morphologic limits, an optimum dental implant size exists for decreasing the stress magnitudes at the bone–implant interface.

In general, the use of short implants has not been recommended because it is believed that occlusal forces must be dissipated over a large implant area for the bone to be preserved. Lum[59] has shown that occlusal forces are distributed primarily to the crestal bone rather than evenly throughout the entire surface area of the implant interface. Because masticatory forces are light and fleeting, these forces are normally well-tolerated by the bone. It is the bruxing forces that must be adequately attenuated, and this may be achieved by increasing the diameter and number of implants. A recent clinical study concluded that short implants are possible when the peri-implant tissues are in good condition.[60]

In summary, the optimum length and diameter necessary for long-term success depends on the bone support condition. If the bone is in normal condition, length and diameter appear not to be significant factors for implant success. However, if the bone condition is poor, large diameter implants are advised and short implants should be avoided.

With regard to implant shape, theoretical analysis implies that clinically, whenever possible, an optimum and not necessarily larger dental implant shape should be used based on the specific morphologic limitations of the mandible.

Holmgren et al[48] reported that a stepped cylindrical design for press-fit situations is most desirable from the standpoint of stress distribution to surrounding bone. With the use of FEA to analyze a parasaggital model digitized from a CT-generated patient data set, these authors simulated various single-tooth, 2-dimensional osseointegrated dental implant models. The results suggested that stress is more evenly dissipated throughout the stepped cylindrical implant than the straight implant type. After analyzing stress concentration patterns using FEA, Rieger et al[56] concluded that a tapered endosseous implant with a high elastic modulus would be most suitable. Also using FEA, Mailath et al[55] compared cylindrical and conical implant shapes exposed to physiologic stresses and examined the occurrence of stress concentrations at the site of implant entry into bone. They reported that cylindrical implants were preferable to conical implant shapes.

Siegele and Soltesz[61] compared cylindrical, conical, stepped, screw, and hollow cylindrical implant shapes by means of FEA. Both a fixed bond (simulating complete load transfer with bioactive materials) and a pure contact (only compression transfer with bioinert materials) without friction between implant and bone were considered interface conditions. The results demonstrated that different implant shapes lead to significant variations in stress distributions in bone. The authors stated that implant surfaces with very small radii of curvature (conical) or geometric discontinuities (stepped) induce distinctly higher stresses than smoother shapes (cylindrical, screw-shaped). Moreover, a fixed bond between implant and bone in the medullary region (as may be obtained with a bioactive coating) is advantageous for the stress delivered to bone because it produces a more uniform stress distribution than does a pure contact.

Patra et al[37] reported that the tapered thread design of the Branemark implant exhibited higher stress levels in bone than the parallel profile thread of the BUD implant (BUD Medical Devices, Inc), which seemed to distribute stresses more evenly. Clift et al[62] reported that the modification of the standard implant design to include a flexible central post resulted in a decrease in the maximum von Mises stresses and equivalent strains in cancellous bone. It was postulated that this would reduce the likelihood of bone fatigue failure and subsequent bone resorption.

Optimum implant shape is related to the bone condition and implant material properties. Implant designs have adopted various shapes; FEA seems to indicate that for commercially pure titanium (cpTi) implants, smoother profiles engender lower stress concentrations. The optimal thread design to achieve the best load transfer characteristics is the subject of current investigations.

Implant surface structure

Bioactive materials are used as coating on titanium implants because they have the potential to encourage bone growth up to the surface of the implant.[63] It is claimed that these coatings can produce a fully integrated interface with direct bonding between bone and the implant material, leading to a more even transfer of load to the bone along the implant and thus a reduction in stress concentrations.[61]

Meijer et al[64] investigated the influence of a 3-layer flexible coating of Polyactive on bone stress distribution with the use of a 3-dimensional FEA in a mandibular model. Polyactive is a system of poly (ethylene oxide) poly (butylene terephthalate) segmented copolymers with bone-bonding capacity. In the case of sagittal and transversal loading, the use of a Polyactive coating reduced both the minimum principal stress in the bone and the compressive radial stress at the bone–implant interface. However, it raised the maximum principal and tensile radial stresses. In the case of vertical loading, the application of a flexible coating reduced the compressive radial stress at the bone–implant interface around the neck of the implant by a factor of 6.6 and the tensile radial stress by a factor of 3.6. Variations in composition and thickness of the coating did not affect the results significantly.

Nature of the bone–implant interface

There are 2 types of contact at the bone–implant interface: bone–implant contact and fibrous tissue–implant contact. The clinical concept of fibrous encapsulation of an implant is considered to be a failure; this condition is no longer modelled in FEA studies.

Surrounding bone quality and quantity

The long-term clinical performance of a dental implant is dependent on the preservation of good quality bone surrounding the implant and a sound interface between the bone and biomaterial. Good quality bone is itself dependent on the appropriate level of bone remodelling necessary to maintain the bone density and the avoidance of bone microfracture and failure. Both processes are governed by the stress and strain distribution in the bone.

Crestal bone: The crestal bone region is of particular interest because of the observations of progressive bone resorption (saucerization). Crestal bone loss is observed around various designs of dental implants. A possible cause of this bone loss is related to the low stresses acting on peri-implant bone. On the basis of both histologic examination and FEA results,[65,66] an equivalent stress of 1.6 MPa has been deemed sufficient to avoid crestal bone loss from disuse atrophy in the canine mandibular premolar region.

Wiskott and Belser[67] studied the relationship between the stresses applied and bone homeostasis of different implant neck designs. It has been observed that the polished neck of dental implants does not osseointegrate as do textured surfaces. Lack of osseointegration was postulated to be due to increased pressure on the osseous bed during implant placement, establishment of a physiologic "biologic width," stress shielding, and lack of adequate biomechanical coupling between the load-bearing implant surface and the surrounding bone. Any viable osseous structure (including the tissue that surrounds the polished implant neck) is subjected to periodic phases of resorption and formation. Hansson[68] compared implants with smooth necks to implants with retention elements all the way up to the crest. His FEA study found that retention elements at the implant neck resulted in a major decrease in peak interfacial shear stresses. He suggests that these retention elements at the implant neck will counteract marginal bone resorption in accordance with Wolff's law.

For the Screw-Vent implant, Clelland et al[69] showed that under axial loading, mesial and distal stresses were much lower than those buccal and lingual to the implant. Maximum stress in the bone was lingual to the superior portion of the collar. Previous longitudinal radiographic studies of a similar implant revealed bone loss mesial and distal to the implant. The authors emphasized that the clinical significance of the stress transfer to the bone buccal and lingual to the implant had yet to be determined.

Minimum required load for avoidance of crestal bone loss appears to have been defined,[37,65,66] but the upper limit of the physiologic stress range has not yet been investigated.

Cortical bone: The quality and quantity of the surrounding bone influences the load transfer from implant to bone.[46,70] In almost all FEA studies of titanium implants, stress concentrations occur around the implant neck. Under oblique loads with high occlusal stress magnitudes, the elastic limit of bone surrounding implants may be surpassed and lead to microfractures in the cortical bone. Clift et al[46] emphasized the importance of having good quality dense bone around the implant neck that can withstand stresses in the range of 9 to 18 MPa before loading. Failure to achieve this after implantation and subsequent healing may result in local fatigue failure and resorption at the neck on resumption of physiologic loading.[71] Holmes and Loftus[72] used FEA to examine the influence of bone quality on the transmission of occlusal forces for endosseous dental implants. Placement of implants in bone with greater thickness of the cortical shell and greater density of the core resulted in less micromovement and reduced stress concentration, thereby increasing the likelihood of fixture stabilization and tissue integration.

With a 3-dimensional FEA model, Papavasiliou et al[73] showed that the absence of cortical bone increased interfacial stresses at the locations studied. Clift et al[71] reported that a reduction, by a factor of 16, in the elastic modulus of the bone around the neck of the implant produced only a 2-fold reduction in the peak stress.

Trabecular (cancellous) bone: Using the degree of direct bone–implant interface as an indicator of endosseous implant success appears to be misleading, as 100% bone apposition is almost never obtained at the surface of the endosseous dental implant. Investigating the 3-dimensional bone interface to hydroxyapatite-coated titanium alloy implants, Wadamoto et al[74] generated computer graphics by the integration of data for serial ground surfaces obtained at 75 µm intervals of the tissue block involved with the implant. The authors found that the bone contact ratio of the whole surface of each of 3 implants was 80.8%, 68.1%, and 68.8%, and the bone contact ratio for each direction and portion varied with the conditions of implant placement. The bone volume ratios around the implant at the 0 to 300 µm zone were also calculated, and total ratios ranged from 58% to 81%. These results may provide useful quantitative information about the bone structure around implants and contribute to the development of more realistic FEA models based on the biologic bone structure around implants.

Patra et al[37] modelled progressive bone loss and partial osseointegration by both 2- and 3-dimensional FEA. When 25%, 75%, and 100% osseointegration was modelled, cortical bone was shown to carry most of the load, with resulting overload leading to crestal bone loss. Stress plots showed that with increasing crestal bone loss, the majority of the load was transferred directly to the weaker trabecular bone tissue.

Clelland et al[75] investigated a Steri-Oss implant in various bone models with different cancellous and cortical bone conditions using 2-dimensional FEA. For the all-cancellous bone model, low stresses and high strains surrounded the implant apex. For the models with a layer of cortical bone added, higher crestal stresses and lower apical strains were observed. The thicker layer of isotropic cortical bone produced stresses at least 50% less than the thinner layer. The assumption of transverse isotropy (orthotropy) for the cortical bone layer increased stresses and strains by approximately 25% compared with isotropic bone. The authors concluded that crestal cortical layer thickness and bone isotropy have a substantial impact on resultant stresses and strains.

Summary

Load transmission and resultant stress distribution are significant in determining the success or failure of an implant. Factors that influence the load transfer at the bone–implant interface include the type of loading, implant and prosthesis material properties, implant length and diameter, implant shape, structure of the implant surface, nature of the bone–implant interface, and quality and quantity of the surrounding bone. Of these biomechanical factors, implant length, diameter, and shape can be changed easily. Cortical and cancellous bone quality and quantity need to be assessed clinically and should influence implant selection.

IMPLANT–PROSTHESIS CONNECTION

Clinical studies have reported a significant incidence of component failure. These include gold screw and abutment screw failures as well as gold cylinder, framework, and implant fractures. The cause of these failures is complex and involves cyclic fatigue, oral fluids, and varied chewing patterns and loads.

Biomechanically, the following component interfaces can be found in the Branemark implant: (1) fixture–abutment interface, (2) abutment screw–abutment interface, (3) gold cylinder–abutment interface, (4) gold retaining screw–gold cylinder interface, and (5) gold retaining screw–abutment screw interface. Long-term screw joint integrity at the implant–abutment screw joint and abutment–gold cylinder screw joint is essential for prosthetic success. An increasing number of FEA studies focus on biomechanical problems involving the screw joint and on screw loosening phenomena.[76-79]

Screw loosening

The screw loosening problem frequently affects dental implants and implant-supported prostheses.

When a screw is fastened to fix the prosthesis, a tensile force (preload) is built up in the shank of the screw. This preload acts on the screw shank from the head of the screw to the threads. The preload should be as high as possible because it creates a clamping force between the abutment and implant. The screw elongates when subjected to tensile forces during tightening. The more elongation there is, the better the stability of the screw in place. Thus, screw design is of significance and should allow maximum torque to be introduced into the shank of the screw.[77]

Several authors[76,77] have drawn attention to the fact that repeated loading and unloading cycles result in alternating contact and separation of components. Clinical findings of screw loosening and failure probably result from these separation events and from elevated strains in the screw. The other mechanism of screw loosening is related to the fact that no surface is completely smooth. Because of the microroughness of components, when the screw interface is subjected to external loads, micromovements occur between the surfaces. Wear of the contact areas may result from these motions, thereby bringing the 2 surfaces closer to each other and causing a decrease in preload in the set of screws.

With prosthesis superstructure distortion, an external preload can be superimposed on the screw joints of the implant prosthesis. This distortion (or lack of passive fit) can impart additional axial forces and bending moments on the screw joints and increase the likelihood of prosthetic component failure.[80]

Application of preload: The load-transfer mechanism between prosthetic components arises from torque application to the abutment screw and gold screw. Sakaguchi et al[76] developed a 2-dimensional FEA model for nonlinear contact analysis of Branemark implant prosthetic components. They found that when the gold retaining screw was fastened into the abutment screw, clamping force on the implant was increased at the expense of a decrease in the clamping force at the abutment screw–abutment interface by 50%. Maximum tensile stresses in the screw after preload were less than 55% of the yield stress.

Cheong et al[81] used FEA to predict that, at a preload tension of 230 N in the gold retaining screw shank, the clamping force at the abutment–abutment screw interface would first be reduced to zero. With further tightening of the gold retaining screw, the rate of increase of stresses in the gold retaining screw was faster than that of the abutment screw; thus, it was predicted that the gold retaining screw would fail first. Failure of the gold retaining screw by yielding was expected for a tensile load of approximately 400 N applied to the gold cylinder. At this 400 N tensile load, the clamping force at the implant–abutment interface was reduced to zero. This affects the overall stability of the implant–prosthesis connection and eventually leads to component failure.

Because preload application to the gold retaining screw reduces the clamping force at the abutment–implant interface, it is recommended that a balance preload be found between the gold retaining screw and abutment screw to make the whole implant–prosthesis connection more stable.[81] The current manufacturer recommendation for the Branemark system is to use tightening torques of 20 Ncm for titanium abutment screws and 10 Ncm for gold retaining screws.

Washer: The addition of a customized washer to dental implant screw joint systems may offer a very simple and inexpensive solution for the persistent problem of screw loosening. With the use of FEA, Versluis et al[79] studied the effect of a washer in a Branemark-type implant on the loosening conditions of the retaining screw. Their simulation indicated that a washer may significantly increase the axial tolerance of a screw against loosening up to 15 times more than a conventional system without a washer. The authors indicated that this is accomplished by increasing the tolerance of the implant against deformation.

Screw fracture

Factors that contribute to screw failure include the magnitude and direction of loading, the elastic modulus of the prosthesis, and the rigidity of the abutment.

By studying the IMZ implant system with FEA, Holmes et al[82] found that with increases in either load magnitude or load angle, stress concentrations in components of the implant system were generally increased. In another study, Holmes et al[83] also showed that in the IMZ implant, stress concentrations in bone and in components were much greater under a 30-degree load than under an equal vertical load. Greater deflection and stress concentrations within the coronal retaining screw were predicted with the use of the resin polyoxymethylene (POM) intramobile element (IME) than with the titanium element in the IMZ implant system. The authors' FEA model also found that stress transmission to bone was not reduced when the IME was modelled in POM rather than titanium. Maximum stress concentrations occurred in the fastening screw.

Several authors[82,84] recommend high elastic modulus prostheses to avoid deflection of the prosthetic superstructure and stress concentrations in the retaining screw. Rigid abutment design is also needed to decrease the peak stresses in the screw and the deflection of the superstructure. Two related studies[85,86] described an FEA model of 3 different IMZ abutment designs: original threaded intramobile element (IME), abutment complete (ABC), and intramobile connector (IMC). Progressive tightening of the retaining screw (preload) was simulated, and the degree of screw tightening necessary to prevent opening of the crown–abutment interface in extreme loading (500 N occlusal load at 45 degrees) was determined individually for each system. A correlation was observed between the peak stresses in the screw and the deflection of the superstructure. Deflections and stress concentrations with the IMC were predicted to be in the same range as with the IME, but much greater than with the ABC.

Summary

The screw loosening problem is of concern, especially when considering single-tooth implant prostheses. The application of optimal preload has been the main means of preventing loosening. However, a recent FEA study advocates the addition of a washer as a simple and effective solution for the persistent problem of screw loosening. Stress concentrations in the fastening screws are influenced by load magnitude and direction. High-rigidity prostheses and rigid abutments have been found to give more favorable stress distributions in the screws.

MULTIPLE-IMPLANT PROSTHESES

From a biomechanical viewpoint, there are 3 main classes of multiple-implant prostheses: (1) implant-supported fixed prostheses (including cantilevered designs), (2) implant-supported overdentures, and (3) combined natural tooth and implant-supported prostheses. FEA studies for these prosthetic situations are usually more complex than for the solitary implant. In most studies, 3-dimensional FEA is considered necessary and 2-dimensional FEA inadequate.

Because multiple implants are splinted by the prosthesis framework, stress distribution is more complex than with the single-tooth implant situation. Loading at one point of the prosthesis causes stress concentrations in all supporting implants to varying degrees. The prosthesis can be loaded not by a single load but by multiple loads and in varying directions. In addition, the flexure of the jaw bones, particularly the mandible, under functional loading conditions can cause stress in the bone around the implants and may lead to bone resorption. Stress around the implant can be caused not only by local deformation of the bone because of movement of the implant and interface relative to the surrounding bone, but also by the complex deformation patterns of the mandible.

Implant-supported fixed prostheses

For implant-supported fixed prostheses, the factors that affect bone-implant stress distribution and ultimately the success of the prosthesis include implant inclination, implant number and position, the prosthesis splinting scheme, the occlusal surface, framework material properties, and different framework cross-sectional beam shapes.

Canay et al[87] compared vertically orientated implants with angled implants and found that the inclination of implants greatly influences stress concentrations around the implant-supported fixed prosthesis. The authors found no measurable differences in stress values and contours when a horizontal load was applied to the vertical and angled implants. However, with vertical loading, compressive stress values were 5 times higher around the cervical region of the angled implant than around the same area in the vertical implant.

Many clinicians are of the opinion that the selection of implant positions and the scheme of prosthesis splinting are critical for the longevity and stability of an implant prosthesis. Kregzde[88] reported that induced stresses in bone are sensitive to the scheme of prosthesis splinting and implant positions. He used 3-dimensional FEA modelling of jaw bones, teeth, and various implant numbers, positions, and prosthesis designs to attempt optimization of stress distribution to the implants. Induced stresses on implants for different schemes of prosthesis splinting and different implant positions were found to vary as much as 1000%.

The effect of different cross-sectional beam configurations for implant frameworks also has been investigated with FEA. Korioth and Johann[89] compared superstructures with different cross-sectional shapes and material properties during a simulated, complex biting task that modelled the deformation patterns of the mandible during function. When they submitted their model to loads mimicking simultaneous bending and torsion of the mandibular corpus during bilateral posterior occlusion, they found that predicted implant stresses varied significantly between implant sites for different superstructure shapes. Contrary to expectations, the ideal "I-beam" superstructure cross-section did not yield the lowest principal stresses; these were obtained with a vertically orientated, rectangular-shaped beam superstructure. The authors concluded that implant abutment stresses were significantly affected by the cross-sectional shape of the prosthetic superstructure and by diverse mandibular loading conditions.

Implant-supported fixed prostheses with cantilevers add additional factors that can influence stress distribution. These factors include cantilever length, cross-sectional beam shapes, and recently, a system for additional support of the distal extension of the cantilever. Young et al[90] investigated a number of different cross-sectional beam shapes for cantilever fixed prostheses for initiation of permanent deformation on end loading. Straight and curved cantilever beams 26 mm long were modelled in FEA. They found that the "L-shaped" design was more rigid than other designs for a given mass and that an open "I-section" framework offered good possibilities, particularly when used as curved shapes. "L-shaped" cobalt-chromium or stainless steel frameworks of 26 mm cantilever span underwent permanent deformation at end loadings between 130 and 140 N, depending on section curvature. The authors caution that good framework design is critical to avoid failures, because it is known that occluding loads can exceed these values.

Different material properties affect stress distribution in different ways. Korioth and Johhann[89] showed that an increase in elastic modulus of prosthetic materials does not necessarily lead to a decrease in stresses on all existing implant abutments. Less rigid superstructures seem to not only increase implant abutment stresses overall but also decrease tensile stresses on the most anterior implant abutments for the modelled complex occluding task.

With the use of a 3-dimensional FEA model and a 6-implant-supported mandibular complete arch fixed prosthesis, Sertgöz[15] investigated the effect of different occlusal surface materials (resin, resin composite, and porcelain) and different framework materials (gold, silver-palladium, cobalt-chromium, and titanium alloys) on stress distribution in the fixed prosthesis and surrounding bone. He demonstrated that the use of a prosthesis superstructure material with lower elastic modulus did not lead to substantial differences in stress patterns or levels in the cortical and cancellous bone surrounding the implants. For the single loading condition investigated, the optimal combination of materials was found to be cobalt-chromium for the framework and porcelain for the occlusal surface.

With a 3-dimensional FEA model of a bilateral distal cantilever fixed prosthesis supported by 6 implants in the mandible, Sertgöz and Guvener[91] predicted that maximum stresses would occur at the most distal bone–implant interface on the loaded side and that these stresses would significantly increase with an increase in cantilever length. Instead, they found no significant change in stress levels associated with implant length variation. In a 15-year longitudinal clinical follow-up study, however, Lindquist et al[92] reported that bone at the distal implants of cantilevered mandibular implant-supported prostheses remained very stable and, conversely, more bone loss was observed around the anterior implants. This result may have been caused by a multitude of other clinical factors. The authors concluded that occlusal loading factors such as maximal occlusal force, tooth clenching, and cantilever length were of minor importance to bone loss in their study population. This suggests that extrapolation of FEA studies to clinical situations should be approached with caution.

New systems for additional support of the distal extensions of cantilevered prostheses have been suggested. The IL system uses a short implant and a special ball-type attachment to support the distal extension of cantilevered prostheses. With the use of 2-dimensional FEA, Lewinstein et al[32] compared the IL support system with a conventional cantilever prosthesis. They concluded that the former dramatically lowered the stresses in the bone, cantilever, and implants and thereby potentially reduced failures within the implants, prosthesis, and surrounding bone. The system also makes possible the employment of a relatively long-span prosthetic extension in the posterior region of the jaw.

In summary, stress distribution in implant-supported fixed prostheses has been shown by FEA to be influenced in various ways by implant inclination, implant number and position, the prosthetic splinting scheme, superstructure material properties, and beam design.

Implant-supported overdentures

The use of implant-supported overdentures is viewed as a cost-effective treatment modality. Some clinicians believe that the designed stress-breaking features of overdenture attachments confer more favorable biomechanical characteristics compared with implant-supported fixed prostheses. Implant-supported overdenture attachment systems include bar-clips, balls, O-rings, and magnets. The biomechanical factors related to bar-clip attachment systems include the number of implants, bar length, stiffener height, and material properties.

Meijer et al[93] set up a 3-dimensional model of a human mandible with 2 endosseous implants in the interforaminal region and compared stress distribution when the 2 implants were connected by a bar or remained solitary. The most extreme principal stress was found with oblique occlusal loads, whereas vertical occlusal loads resulted in the lowest stress. The most extreme principal stresses in bone were always located around the necks of the implants. No significant differences in stress distribution were predicted with the highest maximum and lowest minimum principal stresses being 7.4 and –16.2 MPa in the model without the bar and 6.5 and –16.5 MPa in the model with the bar. The same authors also found that a bar placed anterior to the interconnecting line between the 2 implants caused extremely large compressive and tensile stress concentrations in the bone around the implants. Therefore, in such situations, they advise not connecting the implants or, if a bar-clip attachment is preferred, placing additional implants in the frontal region.[94]

In a later article, Meijer et al[95] used the same model to study a 4-implant system with the implants either connected by a bar or remaining solitary. The results showed that with uniform loading, there were more or less equal extreme principal stresses around the central and lateral implants; with nonuniform loading of the superstructure, the implant nearest the load showed the highest stress concentration. With connected implants, there was a reduction in the magnitude of the extreme principal stresses compared with solitary implants.

In the range of alloy stiffness tested, FEA modelling of a 2-implant round bar[96] and Hader bar system[97] as well as a 4-implant Hader bar system[98] found span length and stiffener height to be more significant factors in the adequacy of the overall design than changing material properties.

Overdentures supported by a 2-implant ball system have been shown to result in better stress distribution in bone than a 2-implant bar system. Menicucci et al[99] used 3-dimensional FEA to evaluate transmission of masticatory load in mandibular implant-retained overdentures. Overdentures retained either by 2 ball attachments or by 2 clips on a bar connecting 2 implants were compared. For the ball attachment system, a 35 N load on the first mandibular molar of the overdenture induced a greater reaction force on the distal edentulous ridge mucosa of the nonworking side than the bar-clip attachment. However, when peri-implant bone stress was considered, such stress was greater with the bar-clip attachment than with the ball attachment.

In summary, FEA has been used to investigate the stress distribution obtained when implants are left solitary, used with ball attachments, or connected by bars for clip retention in various configurations and designs. Not all studies modelled the overdenture over the implants and bar superstructure. Bar design factors like stiffener height and span length were found to significantly affect stress distribution, whereas the influence of various material moduli was comparatively less significant.

Combined natural tooth and implant-supported prostheses

Combining natural teeth and implants to support fixed prostheses has been advocated by certain investigators of implant dentistry. Controversy exists as to the advisability of this design philosophy from a biomechanical as well as a clinical perspective. A significant clinical consideration in the restoration of partial edentulism with implant- and tooth-supported prostheses is whether implants and natural teeth abutments should be splinted, and if so, in what manner. There is a differential deflection between the viscoelastic intrusion of a natural tooth in its periodontal ligament and the almost negligible elastic deformation of an osseointegrated implant. This difference may induce a fulcrum-like effect and possibly overstress the implant or surrounding bone. Some factors that biomechanically influence the stress distribution include abutment design, implant material properties, the effect of resilient elements, connector design (precision or semiprecision attachments), and the degree of splinting implants to natural tooth abutments.

For the implant connected with a natural tooth, van Rossen et al[100] concluded that a more uniform stress was obtained around implants with stress-absorbing elements of low elastic modulus. They also concluded that the bone surrounding the natural tooth showed a decrease in peak stresses in such a situation.

el Charkawi et al[101] studied the use of a resilient layer material under the superstructure of the implant in a connected tooth- and implant-supported prosthesis model. Their FEA proposed that this new modification could mimic the structural natural tooth unit by allowing movement of the superstructure without movement of the implant when the model was loaded.

Misch and Ismail[102] conducted a 3-dimensional FEA comparing models representing a natural tooth and an integrated implant connected by rigid and nonrigid connectors. On the basis of the similarities in stress contour patterns and the stress values generated in both models, the authors concluded that it may be erroneous to advocate a nonrigid connection because of a biomechanical advantage. Melo et al[103] also investigated tooth- and implant-supported prostheses in free-end partially edentulous situations. Their 2-dimensional FEA predicted that lowest levels of stress in bone occurred when the prosthesis was not connected to a natural abutment tooth but instead was supported by 2 freestanding implant abutments. Nonrigid attachments, when incorporated into a prosthesis, did not significantly reduce the level of stress in bone. A recent comprehensive review of both clinical and laboratory studies concluded that the issue of connecting natural teeth to implants with rigid or nonrigid connectors remains unresolved.[104]

CONCLUSIONS

FEA has been used extensively in the prediction of biomechanical performance of dental implant systems. This article reviewed the use of FEA in relation to the bone–implant interface, the implant–prosthesis connection, and multiple-implant prostheses. Assumptions made in the use of FEA in implant dentistry have to be taken into account when interpreting the results.

In modelling, some assumptions greatly affect the predictive accuracy of the FEA model. These include assumptions involving model geometry, material properties, applied boundary conditions, and the bone–implant interface. To achieve more realistic models, advanced digital imaging techniques can be used to model bone geometry in greater detail; the anisotropic and nonhomogenous nature of the material needs to be considered; and boundary conditions must be refined.

In addition, modelling of the bone–implant interface should incorporate the actual osseointegration contact area in cortical bone as well as the detailed 3-dimensional trabecular bone contact pattern.

Load transmission and resultant stress distribution at the bone–implant interface have been the subject of FEA studies. Factors that influence load transfer at the bone–implant interface include the type of loading, implant and prosthesis material properties, implant length and diameter, implant shape, structure of the implant surface, nature of the bone–implant interface, and the quality and quantity of the surrounding bone. Of these biomechanical factors, implant length, diameter, and shape can be modified easily in the implant design. Cortical and cancellous bone quality and quantity need to be assessed clinically and should influence implant selection.

Stress distribution in the implant–prosthesis connection has been examined by FEA studies because of the incidence of clinical problems such as gold and abutment screw failures and implant fracture. Design changes to avoid or reduce these prosthetic failures by improving the stress distribution of implant components have been suggested.

When applied to multiple-implant prosthesis design, FEA has suggested improved biomechanical situations when factors such as implant inclination, implant position, prosthetic material properties, superstructure beam design, cantilever length, bar system, bar span length and stiffener height, and overdenture attachment type are optimized. For combined natural tooth and implant-supported prostheses, FEA studies have not determined conclusively whether rigid or resilient implant systems should be used.

FEA is an effective computational tool that has been adapted from the engineering arena to dental implant biomechanics. With FEA, many design feature optimizations have been predicted and will be applied to potential new implant systems in the future.

REFERENCES

1. Weinstein AM, Klawitter JJ, Anand SC, Schuessler R. Stress analysis of porous rooted dental implants. J Dent Res 1976;55:772-7.
2. Atmaram GH, Mohammed H. Stress analysis of single-tooth implants. I. Effect of elastic parameters and geometry of implant. Implantologist 1983-84;3:24-9.
3. Atmaram GH, Mohammed H. Stress analysis of single-tooth implants. II. Effect of implant root-length variation and pseudo periodontal ligament incorporation. Implantologist 1983-84;3:58-62.
4. Mohammed H, Atmaram GH, Schoen FJ. Dental implant design: a critical review. J Oral Implantol 1979;8:393-410.
5. Borchers L, Reichart P. Three-dimensional stress distribution around a dental implant at different stages of interface development. J Dent Res 1983;62:155-9.
6. Cook SD, Weinstein AM, Klawitter JJ. A three-dimensional finite element analysis of a porous rooted Co-Cr-Mo alloy dental implant. J Dent Res 1982;61:25-9.
7. Meroueh KA, Watanabe F, Mentag PJ. Finite element analysis of partially edentulous mandible rehabilitated with an osteointegrated cylindrical implant. J Oral Implantol 1987;13:215-38.
8. Williams KR, Watson CJ, Murphy WM, Scott J, Gregory M, Sinobad D. Finite element analysis of fixed prostheses attached to osseointegrated implants. Quintessence Int 1990;21:563-70.
9. Akpinar I, Demirel F, Parnas L, Sahin S. A comparison of stress and strain distribution characteristics of two different rigid implant designs for distal-extension fixed prostheses. Quintessence Int 1996;27:11-7.
10. Korioth TW, Versluis A. Modeling the mechanical behavior of the jaws and their related structures by finite element (FE) analysis. Crit Rev Oral Biol Med 1997;8:90-104.
11. Van Oosterwyck H, Duyck J, Vander Sloten J, Van der Perre G, De Cooman M, Lievens S, et al. The influence of bone mechanical properties and implant fixation upon bone loading around oral implants. Clin Oral Implants Res 1998;9:407-18.
12. Rieger MR, Mayberry M, Brose MO. Finite element analysis of six endosseous implants. J Prosthet Dent 1990;63:671-6.
13. Rieger MR, Adams WK, Kinzel GL. Finite element survey of eleven endosseous implants. J Prosthet Dent 1990;63:457-65.
14. Meijer GJ, Starmans FJM, de Putter C, van Blitterswijk CA. The influence of a flexible coating on the bone stress around dental implants. J Oral Rehabil 1995;22:105-11.
15. Sertgöz A. Finite element analysis study of the effect of superstructure material on stress distribution in an implant-supported fixed prosthesis. Int J Prosthodont 1997;10:19-27.
16. Keyak JH, Meagher JM, Skinner HB, Mote CD. Automated three-dimensional finite element modeling of bone: a new method. J Biomed Eng 1990;12:389-97.
17. Cahoon P, Hannam AG. Interactive modeling environment for craniofacial reconstruction. Visual data exploration and analysis. SPIE Proc 1994;2178:206-15.
18. Davy DT, Dilley GL, Krejci RF. Determination of stress patterns in root-filled teeth incorporating various dowel designs. J Dent Res 1981;60:1303-10.
19. Wright KW, Yettram AL. Reactive force distributions for teeth when loaded singly and when used as fixed partial denture abutments. J Prosthet Dent 1979;42:411-6.
20. Farah JW, Hood JA, Craig RG. Effects of cement bases on the stresses in amalgam restorations. J Dent Res 1975;54:10-5.
21. Farah JW, Craig RG, Merouch KA. Finite element analysis of three- and four-unit bridges. J Oral Rehabil 1989;16:603-11.
22. Reinhardt RA, Pao YC, Krejci RF. Periodontal ligament stresses in the initiation of occlusal traumatism. J Periodontal Res 1984;19:238-46.
23. MacGregor AR, Miller TP, Farah JW. Stress analysis of mandibular partial dentures with bounded and free-end saddles. J Dent 1980;8:27-34.
24. Atmaram GH, Mohammed H. Estimation of physiologic stresses with a nature tooth considering fibrous PDL structure. J Dent Res 1981;60:873-7.
25. Rice JC, Cowin SC, Bowman JA. On the dependence of the elasticity and strength of cancellous bone on apparent density. J Biomech 1988;21:155-68.
26. Cook SD, Klawitter JJ, Weinstein AM. A model for the implant-bone interface characteristics of porous dental implants. J Dent Res 1982;61:1006-9.
27. Cowin SC. Bone mechanics. Boca Raton (FL): CRC Press; 1989.
28. Knoell AC. A mathematical model of an in vitro human mandible. J Biomech 1977;10:159-66.
29. Maeda Y, Wood WW. Finite element method simulation of bone resorption beneath a complete denture. J Dent Res 1989;68:1370-3.
30. Sakaguichi RL, Borgersen SE. Nonlinear contact analysis of preload in dental implant screws. Int J Oral Maxillofac Implants 1995;10:295-302.
31. Colling EW. The physical metallurgy of titanium alloys. Metals Park (OH): American Society for Metals; 1984.
32. Lewinstein I, Banks-Sills L, Eliasi R. Finite element analysis of a new system (IL) for supporting an implant-retained cantilever prosthesis. Int J Oral Maxillofac Implants 1995;10:355-66.
33. Craig RG. Restorative dental materials. 8th ed. St. Louis (MO): Mosby; 1989. p. 84.
34. Lewis G. Aparametric finite element analysis study of the stresses in an endosseous implant. Biomed Mater Eng 1994;4:495-502.
35. Lotz JC, Gerhart YN, Hayes WC. Mechanical properties of metaphysical bone in the proximal femur. J Biomech 1991;24:317-29.
36. Cowin SC. Strain assessment by bone cells. Tissue Eng 1988:181-6.
37. Patra AK, DePaolo JM, D'Souza KS, DeTolla D, Meenaghan MA. Guidelines for analysis and redesign of dental implants. Implant Dent 1998;7:355-68.
38. Zhou X, Zhao Z, Zhao M, Fan Y. The boundary design of mandibular

model by means of the three-dimensional finite element method. West China J Stomatol 1999;17:1-6.
39. Teixeira ER, Sato Y, Shindoi N. A comparative evaluation of mandibular finite element models with different lengths and elements for implant biomechanics. J Oral Rehabil 1998;25:299-303.
40. Sato Y, Teixeira ER, Tsuga K, Shindoi N. The effectiveness of a new algorithm on a three-dimensional finite element model construction of bone trabeculae in implant biomechanics. J Oral REhabil 1999;26:640-3.
41. Pilliar RM, Deporter DA, Watson PA, Valiquette N. Dental implant design—effect on bone remodeling. J Biomed Mater Res 1991;25:467-83.
42. Vaillancourt H, Pillar RM, McCammond D. Factors affecting crestal bone loss with dental implants partially covered with a porous coating: a finite element analysis. Int J Oral Maxillofac Implants 1996;11:351-9.
43. Hassler CR, Rybicki EF, Cummings KD, Clark LC. Quantitation of compressive stress and its effects on bone remodeling [proceedings]. Bull Hosp Bone Joint Dis 1977;38:90-3.
44. Skalak R. Biomechanical considerations in osseointegrated prostheses. J Prosthet Dent 1983;49:843-8.
45. Ma Xuanxiang LT. Single implant prosthesis. In: Geng Jianping, editor. Newly developed technology of prosthetic dentistry in China in 1990s. Chengdu: Sichuang Science and Technology Publishing House; 1998. p. 1-20.
46. Clift SE, Fisher J, Watson CJ. Finite element stress and strain analysis of the bone surrounding a dental implant: effect of variations in bone modulus. Proc Inst Mech Eng [H] 1992;206:233-41.
47. Geng J, Liu H. Exceptional prosthodontics. Hong Kong: Hong Kong Tranfor Publishing Co Ltd; 1999. p. 60-76.
48. Holmgren EP, Seckinger RJ, Kilgren LM, Mante F. Evaluating parameters of osseointegrated dental implants using finite element analysis—a two-dimensional comparative study examining the effects of implant diameter, implant shape, and load direction. J Oral Implantol 1998;24:80-8.
49. Barbier L, Vander Sloten J, Krzesinski G, Schepers E, Van der Perre G. Finite element analysis of non-axial versus axial loading of oral implants in the mandible of the dog. J Oral Rehabil 1998;25:847-58.
50. 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.
51. Benzing UR, Gall H, Weber H. Biomechanical aspects of two different implant-prosthetic concepts for edentulous maxillae. Int J Oral Maxillofac Implants 1995;10:188-98.
52. 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.
53. Hobkirk JA, Psarros KJ. The influence of occlusal surface material on peak masticatory forces using ossointegrated implant-supported prostheses. Int J Oral Maxillofac Implants 1992;7:345-52.
54. Cibirka RM, Razzoog ME, Lang BR, Stohler CS. Determining the force absorption on quotient for restorative materials used in implant occlusal surfaces. J Prosthet Dent 1992;67:361-4.
55. Mailath G, Stoiber B, Watzek G, Matejka M. Bone resorption at the entry of osseointegrated implants—a biomechanical phenomenon. Finite element study [in German]. Z Stomatol 1989;86:207-16.
56. Rieger MR, Fareed K, Adams WK, Tanquist RA. Bone stress distribution for three endosseous implants. J Prosthet Dent 1989;61:223-8.
57. Stoiber B. Biomechanical principles of endosseous screw implants [in German]. Wien Klin Wochenschr 1988;100:522-4.
58. Matsushita Y, Kitoh M, Mizuta K, Ikeda H, Suetsugu T. Two-dimensional FEM analysis of hydroxyapatite implants: diameter effects on stress distribution. J Oral Implantol 1990;16:611.
59. Lum LB. A biomechanical rationale for the use of short implants. J Oral Implantol 1991;17:126-31.
60. Stellingsma C, Meijer HJ, Raghoebar GM. Use of short endosseous implants and an overdenture in the extremely resorbed mandible: a five-year retrospective study. J Oral Maxillofac Surg 2000;58:382-7; discussion 387-8.
61. Siegele D, Soltesz U. Numerical investigations of the influence of implant shape on stress distribution in the jaw bone. Int J Oral Maxillofac Implants 1989;4:333-40.
62. Clift SE, Fisher J, Edwards BN. Comparative analysis of bone stresses and strains in the Intoss dental implant with and without a flexible internal post. Proc Inst Mech Eng [H] 1995;209:139-47.
63. Oonishi H. Mechanical and chemical bonding of artificial joints. Clin Mater 1990;5:217-33.
64. Meijer GJ, Starmans FJ, de Putter C, van Blitterswijk CA. The influence of a flexible coating on the bone stress around dental implants. J Oral Rehabil 1995;22:105-11.
65. Vaillancourt H, Pilliar RM, McCammond D. Finite element analysis of crestal bone loss around porous-coated dental implants. J Appl Biomater 1995;6:267-82.
66. Vaillancourt H, Pilliar RM, McCammond D. Factors affecting crestal bone loss with dental implants partially covered with a porous coating: a finite element analysis. Int J Oral Maxillofac Implants 1996;11:351-9.
67. Wiskott HW, Belser UC. Lack of integration of smooth titanium surfaces: a working hypothesis based on strains generated in the surrounding bone. Clin Oral Implants Res 1999;10:429-44.
68. Hansson S. The implant neck: smooth or provided with retention elements. A biomechanical approach. Clin Oral Implants Res 1999;10:394-405.
69. Clelland NL, Ismail YH, Zaki HS, Pipko D. Three-dimensional finite element stress analysis in and around the Screw-Vent implant. Int J Oral Maxillofac Implants 1991;6:391-8.
70. Lum LB, Osier JF. Load transfer from endosteal implants to supporting bone: an analysis using statics. Part one: Horizontal loading. J Oral Implantol 1992;18:343-8.
71. Clift SE, Fisher J, Watson CJ. Finite element stress and strain analysis of the bone surrounding a dental implant: effect of variations in bone modulus. Proc Inst Mech Eng [H] 1992;206:233-41.
72. Holmes DC, Loftus JT. Influence of bone quality on stress distribution for endosseous implants. J Oral Implantol 1997;23:104-11.
73. Papavasiliou G, Kamposiora P, Bayne SC, Felton DA. Three-dimensional finite element analysis of stress-distribution around single tooth implants as a function of bony support, prosthesis type, and loading during function. J Prosthet Dent 1996;76:633-40.
74. Wadamoto M, Akagawa Y, Sato Y, Kubo T. The three-dimensional bone interface of an osseointegrated implant. I: a morphometric evaluation in initial healing. J Prosthet Dent 1996;76:170-5.
75. Clelland NL, Lee JK, Bimbenet OC, Gilat A. Use of an axisymmetric finite element method to compare maxillary bone variables for a loaded implant. J Prosthodont 1993;2:183-9.
76. Sakaguchi RL, Borgersen SE. Nonlinear finite element contact analysis of dental implant components. Int J Oral Maxillofac Implants 1993;8:655-61.
77. Jorneus 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.
78. Haack JE, Sakaguich RL, Sun T, Coffey JP. Elongation and preload stress in dental implant abutment screws. Int J Oral Maxillofac Implants 1995;10:529-36.
79. Versluis A, Korioth TW, Cardoso AC. Numerical analysis of a dental implant system preloaded with a washer. Int J Oral Maxillofac Implants 1999;14:337-41.
80. Tan KBC. The clinical significance of distortion in implant prosthodontics: is there such a thing as passive fit? Ann Acad Med Singapore 1995;24:138-57.
81. Cheong WM. FEA of the Nobel Biocare standard abutment during preload and applied axial loads. [BEng Thesis.] Singapore: National University of Singapore; 2000.
82. Holmes DC, Haganman CR, Aquilino SA. Deflection of superstructure and stress concentrations in the IMZ implant system. Int J Prosthodont 1994;7:239-46.
83. Holmes DC, Grigsby WR, Goel VK, Keller JC. Comparison of stress transmission in the IMZ implant system with polyoxymethylene or titanium intramobile element: a finite element stress analysis. Int J Oral Maxillofac Implants 1992;7:450-8.
84. Papavasiliou G, Tripodakis AP, Kamposiora P, Strub JR, Bayne SC. Finite element analysis of ceramic abutment-restoration combinations for osseointegrated implants. Int J Prosthodont 1996;9:254-60.
85. Holmes DC, Haganman CR, Aquilino SA, Diaz-Arnold AM, Stanford CM. Finite element stress analysis of IMZ abutment designs: development of a model. J Prosthodont 1997;6:31-6.
86. Haganman CR, Holmes DC, Aquilino SA, Diaz-Arnold AM, Stanford CM. Deflection and stress distribution in three different IMZ abutment designs. J Prosthodont 1997;6:110-21.
87. Canay S, Hersek N, Akpinar I, Asik Z. Comparison of stress distribution around vertical and angled implants with finite-element analysis. Quintessence Int 1996;27:591-8.

88. Kregzde M. A method of selecting the best implant prosthesis design option using three-dimensional finite element analysis. Int J Oral Maxillofac Implants 1993;8:662-73.
89. Korioth TW, Johann AR. Influence of mandibular superstructure shape on implant stresses during simulated posterior biting. J Prosthet Dent 1999;82:67-72.
90. Young FA, Williams KR, Draughn R, Strohaver R. Design of prosthetic cantilever bridgework supported by osseointegrated implants using the finite element method. Dent Mater 1998;14:37-43.
91. Sertgöz A, Güvener S. Finite element analysis of the effect of cantilever and implant length on stress distribution in an implant-supported fixed prosthesis. J Prosthet Dent 1996;76:165-9.
92. Lindquist LW, Carlsson GE, Jemt T. A prospective 15-year follow-up study of mandibular fixed prostheses supported by ossointegrated implants. Clinical results and marginal bone loss. Clin Oral Implants Res 1996;7:329-36.
93. Meijer HJ, Starmans FJ, Steen WH, Bosman F. A three-dimensional, finite-element analysis of bone around dental implants in an edentulous human mandible. Arch Oral Biol 1993;38:491-6.
94. Meijer HJ, Starmans FJ, Steen WH, Bosman F. Location of implants in the interforaminal region of the mandible and the consequences for the design of the superstructure. J Oral Rehabil 1994;21:47-56.
95. Meijer HJ, Starmans FJ, Steen WH, Bosman F. Loading conditions of endosseous implants in an edentulous human mandible: a three-dimensional, finite-element study. J Oral Rehabil 1996;23:757-63.
96. Bidez MW, Chen Y, McLoughlin SW, English CE. Finite element analysis (FEA) studies in 2.5-mm round bar design: the effects of bar length and material composition on bar failure. J Oral Implantol 1992;18:122-8.
97. Bidez MW, McLoughlin SW, Chen Y, English CE. Finite element analysis of two-abutment Hader bar designs. Implant Dent 1993;2:107-14.
98. Bidez MW, Chen Y, McLoughlin SW, English CE. Finite element analysis of four-abutment Hader bar designs. Implant Dent 1993;2:171-6.
99. Menicucci G, Lorenzetti M, Pera P, Preti G. Mandibular implant-retained overdenture: finite element analysis of two anchorage systems. Int J Oral Maxillofac Implants 1998;13:369-76.
100. van Rossen IP, Braak LH, de Putter C, de Groot K. Stress-absorbing elements in dental implants. J Prosthet Dent 1990;64:198-205.
101. el Charkawi HG, el Wakad MT, Naser ME. Modification of osseointegrated implants for distal-extension prostheses. J Prosthet Dent 1990;64:469-72.
102. Misch CM, Ismail YH. Finite element stress analysis of tooth-to-implant fixed partial denture designs. J Prosthodont 1993;2:83-92.
103. Melo C, Matsushita Y, Koyano K, Hirowatari H, Suetsugu T. Comparative stress analyses of fixed free-end osseointegrated prostheses using the finite element method. J Oral Implantol 1995;21:290-4.
104. Gross M, Laufer BZ. Splinting osseointegrated implants and natural teeth in rehabilitation of partially edentulous patients. Part I: laboratory and clinical studies. J Oral Rehabil 1997;24:863-70.

Reprint requests to:
Dr Keson B. C. Tan
Department of Restorative Dentistry
Faculty of Dentistry
National University of Singapore
119074 SINGAPORE
Fax: (65) 773-2603
E-mail: rsdtanbc@nus.edu.sg

0022-3913/2001/$35.00 + 0. **10/1/115251**

doi:10.1067/mpr.2001.115251

New product news

The January and July issues of the Journal carry information regarding new products of interest to prosthodontists. Product information should be sent 1 month prior to ad closing date to: Dr. Glen P. McGivney, Editor, UNC School of Dentistry, 414C Brauer Hall, CB #7450, Chapel Hill, NC 27599-7450. Product information may be accepted in whole or in part at the discretion of the Editor and is subject to editing. A black-and-white glossy photo may be submitted to accompany product information.

Information and products reported are based on information provided by the manufacturer. No endorsement is intended or implied by the Editorial Council of *The Journal of Prosthetic Dentistry,* the editor, or the publisher.