A STEP BY STEP GUIDE TO FINITE ELEMENT ANALYSIS IN DENTAL IMPLANTOLOGY

[1]FAISAL MOEEN, BDS, MSc (Singapore)

[2]SALEHA NISAR, BDS

[3]NIMRA DAR, BDS

ABSTRACT

As a consequence of dental implant treatment being considered effective, safe and predictable they are now a preferred solution for those who have lost teeth due to dental caries, periodontal disease, injuries or other reasons. The biomechanical performance of implants cannot be proven or tested in vivo due to difficulties in assessing or quantifying the level of osseointegration, the stability of the implant and various other factors. Finite element analysis (FEA) has been used extensively to examine intractable and inaccessible interfaces of the dental implant assembly and to predict biomechanical performance as well as the effect of clinical factors on implant success. Although an in depth understanding of the theory, method, application and limitations of FEA in implant dentistry will help the clinician to interpret results of FEA studies and extrapolate these results to clinical situations, this article explains the basic practical steps involved in the utilization of FEA from a viewpoint of a researcher interested in conducting studies in dental implantology.

Key Words: *Finite Element Analysis, Modeling, Dental Implants.*

INTRODUCTION

With the acceptance of osseointegration as a reliable and predictable phenomenon, the utilization of dental implants as a long term treatment modality to replace missing teeth is on the rise. From a biomechanical point of view if the stresses imparted on a functioning implant unit and its supporting soft and hard tissues are beyond their adaptive capacity, these would respond either in early or delayed catastrophic breakage of the non-living components or degenerative changes in the peri-implant living tissues. The magnitude, direction and duration of load employed on the restoration-implant apparatus plays a pivotal role in the dissipation of forces from the restoration, abutment, screw, fixture unit into the surrounding bone.[1] Biomechanical failures present as rapid bone loss, infection of the peri-implant soft tissue, breakage of the ceramic or polymeric material of the restoration, loosening of the implant screw and even fracture of the screw, the abutment and in rare cases the entire implant itself.

Randomized control trials are considered the optimal methodology to investigate the performance of the materials and the biomechanical variables of dental implants.[2] These however, are difficult to keep viable for long durations primarily due to attrition of patients and the fact that they might not remain economically feasible. In such a situation, well designed invitro tests such as the Finite Element Analysis or FEA can give valuable information to researchers into the otherwise inaccessible, intractable areas of the dental implant assembly. Although, the use of FEA in dental implantology has been extensive, each step involved in this process is worthy of mention from a practical perspective, in order to facilitate individuals newly entering this research area.

Creating a model

1.1 2D Vs 3D

The quality of the FEA is directly dependant on the models which are to be used in the research. With respect to dimensions and material properties the model should look like the actual structure. The decision on forming 2D or 3D models depends on the intricacy of the problems which are to be addressed, the level of accuracy required, applicability of the results and the complexity of the structures involved in the analysis.[3] It is generally believed that 3D models are more realistic and hence represent the biomechanical interactions of the human anatomy, restorations and implant components as a complex more superiorly than 2D models. Such an advantage however comes with an increase in the level of difficulty involving CAD modeling, solving and output interpretation in comparison to 2D models.[5]

1 Associate Professor of Dental Materials, Islamic International Dental College, Riphah University. G-7/4, Islamabad. Pakistan. Email: faisalmoeen_is@yahoo.com, Cell: 0300-8546021

2 & 3 Scholars of MPhil Dental Materials. Islamic International Dental College, Riphah University. G-7/4, Islamabad. Pakistan

Received for Publication: January 2, 2014
Revision Received: February 10, 2014
Revision Accepted: February 12, 2014

2D FEA has been extensively used in solving mechanical related dentistry problems in the past, such as when qualitatively comparing the results of different cases.[4] They are however limited in the level of accuracy and reliability in comparison to their 3D counterparts and since the general direction of such experiments is towards simulating data as close to clinical reality as possible, 2D FEA have fallen slightly out of favor. Although, 3D models involve a higher level of difficulty in mesh refinements, they are in contrast superior in the level of accuracy while capturing the geometry of complex structures.

3D models can be manually constructed or generated from imaging options such as a CT scan or an MRI. With computed tomography, realistic anatomic features can be modeled along with the inclusion of material properties such as bone density values.[6] The choice for building a model using either a manual or automatic technique depends on the purpose of the study and the structure of interest. The manual input technique to generate 3D structures uses appropriate aided design soft wares such as AutoCAD (Autodesk Inc, San Rafael, CA, USA), Solid Works (Solid Works Corp., Concord, MA, USA), Pro/Engineer (Wildfire, PTC, Needham, MA, USA), Rhino 3D (McNeel North America, Seattle, WA, USA).

1.2 Geometry of a model

The initial step for FEA modeling involves the computerized representation of the geometry. Certain 2D FEA studies modeled the bone simply in a rectangular ellipsoid or a U-shaped configuration with the implant.[7] Specimens from human cadavers can be subjected to CT scans and the images acquired can be used for the 2D or 3D modeling procedure.[8]

If the study involves analysis of only a particular area of the upper or lower jaw, then modeling of the entire anatomical structure is not required, on the contrary this would substantially increase the time, energy and cost associated with a little gain in efficacy of the output. The region of interest which is to be modeled depends on the objectives of the study and be extracted by a number of methods such as the Boolean process. The usual regions of interest involving FEA of dental implants include segments of the mandible, maxilla, maxillary sinus and the temporomandibular joint.[6]

If weak bone, such as that seen in the posterior maxillary region is to be simulated the cortical bone can be neglected altogether. The properties of bone related to density are calibrated to range from soft to dense bone depending on the research protocol; however most studies assume a uniform density value for cortical and cancellous bone. Cancellous bone however is not uniform and is anisotropic and has variation in densities which would affect the magnitude and stress concentrations after loading. Assumptions such as these are common to FEA and are utilized to reduce computing difficulties.[9] Future studies should have the provision of using bone density values measured in Hounsfield Units or from other data obtained from patient specific scans.[10]

When advanced imaging data is used to create a 3D model, the object can automatically be created in the form of masks by thresholding the region of interest on the entire stack of scans (Mimics 13.0, Materialize, and Leuven, Belgium). The degree of automation and high resolution make this model creation method attractive, but determining the appropriate thresholding algorithms to the bone-air boundary reliably throughout a structure with varying bone thicknesses and density can be challenging.[11] Surface smoothing in this approach is advised in order to decrease the number of nodes and elements in the discretized FE model which decreases computation time.[12] However, it is also advisable that when surface smoothing is performed it does not over simplify the geometries, causing a decrease in solution accuracy.

Identifying bone and implant properties

Material properties which include those of living structures and mechanical non-living entities such as implant fixtures, abutments and restorations greatly influence the stress and strain distributions. FEA can be used to model these properties as anisotropic, isotropic, transversely isotropic and orthotropic. For a FEA to generate results which are clinically relevant and as close to how they appear in real life, features that directly affect model accuracy such as interface, loading conditions and material properties should not be neglected or ignored. However in most cases, researchers overlook one or more features in their studies and the majority of previous FE scientific communications in implant prosthodontics have considered material properties to be isotropic, homogeneous, and linear elastic.[13,14, and 4]

An isotropic material indicates that the mechanical response is similar regardless of the stress field direction, requiring Young's modulus (E) and Poisson's ratio (ν) values for the FE calculation. The elastic, or Young's modulus (E), is defined as stress/strain (σ/ε) and is measured in simple extension or compression. It is a measure of material deformation under a given axial load. Poisson's ratio (ν) is the lateral strain divided by axial strain, thus representing how much the sides of a material deform as it is tested.

Bone is an anisotropic material with properties being directionally dependent. Anisotropy can be defined as a difference, when measured along different axes, in a material's physical or mechanical properties such as absorbance, refractive index, conductivity, tensile strength, etc. To incorporate realistic material for bone tissues in maxilla or mandible, the FEM may employ full orthotropy for cortical bone as the elastic behavior in cortical bone approximates to an orthotropic materials and transversely isotropic for cancellous bone. Orthotropy is a form of anisotropy in which the internal configuration of the material results in unique elastic behavior along the three orthogonal axis of the material. In this case, three elastic (E) and shear modulus (G) and six Poisson's ratios (ν) are necessary for model input.

In assuming cancellous bone to be transversely isotropic brings us one step closer to simulating realistic bone properties and demonstrates the significance of using anisotropy. This would also be convenient for the researcher as the material measurement properties for human mandible and maxilla are currently available. A transversely isotropic material behaves identically in all planes perpendicular to the axis of symmetry. This inimitable symmetry axis for cortical bone is along the mesio-distal direction with the bucco-lingual plane being the plane of elastic isotropy. The unique symmetry axis for cancellous bone of edentulous mandible is in the infero-superior direction with the anatomic transverse plane being a plane of elastic isotropy.[15]

To date, no consensus regarding the mechanical properties that are appropriate for simulating the different bone density scenarios clinically encountered in implant dentistry has been reached. For instance, the value of trabacular bone elastic modulus observed in the literature range from 0.3 to 9.5 GPA.16,17 A different approach has been employed by Tada and coworkers who assigned different elastic moduli to bone depending on its density from most dense (Type 1) to least dense (Type IV).[18] For cortical bone, studies typically use the elastic modulus (E) of 13.7 GPa and the Poisson ratio (ν) similar to 0.3 for the trabacular and cortical bone.[13,16]

Non-living components can be modeled by one of two options; Implant and abutment components can either be scanned and digitally reconstructed which are imported into the FE module or these can be manually drawn from precise geometric measurements acquired from the manufactures. The accuracy of the non-living components should ideally be modeled as close to reality as possible, however, this decision depends on the goals of the FE exercise. If the objective of the study is to investigate the magnitude and distribution of stresses created in bone upon placement of a root formed tapered implant, the implant with features such as diameter, taper, length, macro-micro thread configuration needs to be precisely modeled. If the implant fixture and the abutment are modeled separately then a contact condition at a predefined pre-load and torque value, such as the coefficient of friction needs to be set between the two bodies.

Restorations such as single unit crowns can be digitally constructed manually with guidance in respect to mesio-distal, bucco-lingual dimensions from an anatomy atlas. Realistic or even patient specific shapes of crowns and bridges can be modeled from intraoral CAD scans, re-engineering from rapid prototyping and even from dimensions acquired from 3D prints.[35]

Deciding on the bone-implant interface

The original Brånemark protocol on loading prescribed to wait till healing matures and a definitive 'bond' has formed at the implant bone interface. One of the most crucial aspects of FEA is to decide on the conditions to be modeled at the bone-implant interface.[19]

A typical dental implant currently available is not press fit rather is screw turned with the threads of the implant biting into the surrounding walls of the bone, thereby further increasing the seating fixation, a factor measured as insertion torque in Ncm. Depending on the situation, clinicians tend to load the dental implant prior to the completion of osseointegration, immediately after implant placement. Such a loading protocol is termed as 'immediate loading' and has proven to be a reliable treatment decision in favorable circumstances. Frictional contact elements are used to simulate a non-integrated bone to implant interface (i.e. in immediately loaded protocols), which allows minor displacements between the implant and the bone. The occurrence of relative motion between implant and bone introduces a source of non-linearity in FEA, since the contact conditions will change during load application. The frictional coefficient when modeling immediate loading scenarios is usually set at 0.3, such that the contact zone transfers friction and pressure but no tension.[9]

Previous FEA studies employed linear static models with the assumption that bone and implant are perfectly bonded to each other.[20] This assumption is supported by experimental investigations which show fractures occurring further away from the bone-implant interface on removing osseointegrated dental implants with roughened surfaces. In reality, dental implants never have a 100% surface area perfectly bonded to the surrounding bone.[21,22]

FEA softwares should have the provision of using different types of contact algorithms capable of simulating more realistic bone-implant contact types. Three different contact types defined in ANSYS—"bonded", "no separation", and "frictionless" vary not just between patients but can be different in the same patient, such as, are useful to describe the integration at the implant bone interface.[23]

FEA studies which investigate the effect of loading on the peri-implant remodeling of bone cater to the understanding that at a certain minimum threshold of strain energy bone apposition would take place and beneath a lower threshold bone loss would occur due to disuse atrophy. Frost studies indicate that strains in the range of 50-1500 micro strain stimulate cortical bone mass and represent the healthy, bone deposition range.[24] Strain beyond this range and above a critical level may cause overload and hence lead to a loss of bone as the forces imparted are above the resiliency of the attachment.[25]

Another assumption made in FE models is to consider perfect bonding between the implant fixture and the abutment. As the abutment is held inside the fixture by a screw at a certain pre-load and the components have anti-rotational elements, a non-linear contact analysis would be a more realistic method to simulate the micro-motions which can potentially occur between the components. A frictional coefficient of 0.5 is usually assumed between the implant, abutment and screw in FE studies simulating implant-abutment contacts.26 Factors which result in screw loosening i.e. abutment fixture instability include inadequate screw pre-load, over loading, poor fitting of components and elasticity of bone.[27]

Setting boundary conditions

Under ideal circumstances, the entire jawbone structure should be modeled along with the ligaments, muscles, tendons and other supporting tissue to precisely simulate the collective force transmitting unit on a particular implant assembly. This however, is not a pragmatic approach and for the sake of keeping the computations practical the model is kept smaller yet accurate enough to generate correct information. Since the distribution of stress and strain are effective in the region of loading only and if for example the objective of the study is to assess the stresses created in only the peri-implant region or the cortical one third in particular, the entire quadrant of bony segment need not be modeled.28 In addition, Teixeira et al. demonstrated by a 3D FEA that modeling of the mandible greater than 4.2mm mesially or distally from the implant did not result in any significant improvement in accuracy.[29] For the periphery of the model to guarantee an equilibrium solution it is imperative that the boundaries with zero displacement or rotation should be kept at nodes which are at a reasonable distance away from the region of interest so that there is no overlapping between the stress and stain fields associated with the induced reaction forces.

Selecting the loading conditions

Loading can be axial and non-axial. An axial force flows down the long axis of the implant and hence compresses the anchorage unit which is favorable. Non-axial or horizontal loading transmits tensile stresses which try to separate the components and induces a bending movement which is considered destructive. When a crown is to be fabricated for an implant, special attention is given to produce favorable biomechanical forces. Basic engineering principals are to be followed which result in creating a restoration transmitting favorable compressive stresses along the long axis, avoid creating sharp cuspal interfering anatomy and any cantilevers which can result in torsional and bending movement. The combination of axial and non-axial loading, termed as mixed loading, simulates practical conditions where the actual applied force may be inclined with respect to the implant axis and components can be solved in the longitudinal and transverse directions.[30] These oblique forces have been considered more clinically realistic in FEA than vertical ones[31] ranging from one area of the mouth to another.

Bite force studies reveal that magnitude of forces vary from one area of the mouth to another. There are variations related to gender, muscle size and tonicity, parafunctional habits, age and degree of edentulism. In the premolar region, reported values of bite force range from 40-600 N.[32] Average forces of between and up to 50-400 N for young adults have been recorded in the molar region. Small forces of 25 to 170N have been measured in the incisal region.[32] Clinical studies reveal that the average bite force transmitted to endosseous implant range between 90-280N, depending on the location, diameter, length of the implant and the kind of abutment used.[33] These bite force ranges have been used in various FEA investigations.[34]

In FEA literature, the point of loading changes in accordance to the modeled morphology of the restoration.[35] FEA studies have loaded the cuspal tips, distal and mesial fossae of the crowns with the objective of simulating the contact path followed by the functional cusps of a particular tooth. For a proper assessment of stress concentrations is it favorable to consider realistic simulation of biting and hence the loading forces should be applied to the restoration initially and then transmitted through the abutment to the implant and surrounding bone. Finally loading can be static or dynamic. Dynamic loading although more realistic, has been more difficult to computationally model than static loading and hence most FEA use static loads which can be axial, non-axial or mixed.[36]

Convergence testing

In FEA, each finite element has an equation that defines its stiffness [K] based on the physical properties, material and geometry of the element. The stiffness [K] describes the relationship between applied forces {F} and displacements {d} using the basic equation: **{d}={F}/[K]**

As a result, the set of finite elements would have a set of equations and these are connected to each other by constraint equations or equilibrium equations which ensure continuity across the mesh, such that force displacement between elements is equal. The equations used by FEA for solving a problem can be linear or till up to ninth order. Linear equations would result in linear displacements and the strain remains constant. If a higher level of accuracy is needed to simulate displacement within each element much higher usage of equations would be needed (p-method). However, linear element models can be made accurate by adding more elements to the mesh hence resulting in smaller, more precise displacements and stress fields to be generated (h-method). The user therefore, must run the models several times to show evidence of convergence between a numbers of mesh densities for comparison.

In FEA involving implant dentistry where a 3D model is to be created involving irregular shaped objects refining the mesh to achieve convergence might be a difficult task.[37] In such a case a reasonable mesh is created in the p-method and the polynomial level is increased from two to as high as nine as mentioned above. Such a model is then run a few times and considered converged till the global strain energy changes by less than 1%.[38]

The Analysis

The different ways utilized in FE studies to apply a force or moment include load concentrations at a single point or a particular node, forces along a line, interface or an edge, bending and torque.[39] The results obtained from a FEA give a description of the load transfer i.e. stress distribution within components and into the surrounding bone. As compared to an Invitro test, which usually culminates in revealing a maximum load to failure for the system being tested, FEA provides an insight into the actual process which leads to failure.

In reality, the implementation of the results of a FEA into a clinical scenario would be difficult. Having said this, the aim of the exercise should be to mimic the real situation as closely as possible. If the models utilize structures as isotropic and homogenous, the results of the FEA should be interpreted with caution as far as their applicability to a clinical situation is concerned.

Values from Finite Element Analysis are usually presented as either von Mises stress, maximum and minimum principal stresses or maximum and minimum principal strains. Most of the previously published studies have used von Mises stress as an analysis criterion which usually deals with ductile materials having equal compressive and tensile strength such as aluminum or steel. However when representing brittle materials such as bone, ceramics or cements maximum principal stress would better indicate the magnitude of stress concentrations and the distributions as this offers the option of distinguishing between tensile and compressive stresses by positive and negative signs respectively.[40] Also, displacement points give indications of the deformations and help in interpretation of the results. Maximum stress values generated from a FEA can be compared with the ultimate compressive and ultimate tensile strengths of a material.[40] However, when the titanium components such as abutment, screw and implant are being examined which are ductile structures; $\sigma\nu M$ is a recommended analysis criterion and can be used to determine the strain energy.[41] The von Mises stress criteria refers to a formula for combining the stresses acting in the x, y and z dimensions in an equivalent value.

Validate the FE results

It is important to validate the results generated from the FE models, especially when they seem to have clinical/biological implications.[42] A validation is done by comparing the results of the FEA with data available in literature on the subject matter. Validation leads to the confidence that the right models were constructed and whether or not the assumptions and approximations were justified. Does the investigation turn out results which corroborate with authentic information as seen in literature or does it contradict them? One of the established ways of validating results from FEA is to conduct parallel invitro or vivo experiments on the same study matter.[42] If the FE analysis sets out in the beginning with clear objectives to establish a deeper understanding in a novel field with at most to identify a potential trend and does not claim to produce absolute answers, the results can be recommended to be validated in future control trials.

CONCLUSION

With improvements in computer technology and a deeper insight into the theory, methodology, advantages and limitations of FEA, the clinician can utilize this powerful tool in developing a better understanding of the biomechanics of dental implantology. This article has attempted to address the basics of FEA in dental implantology from a practical viewpoint. The ingredients which make FEA a tool powerful enough to reliably comment on versatile stress states in a complex structure are known. Like any other tool used to solve a problem, the solution generated can only be as strong as the proper utilization of the tool itself. Future research should attempt to correlate results with clinical findings thereby increase validity of the models, simulate the effect of saliva, infection and fatigue failure under repetitive, realistic, cyclic loading conditions.

REFERENCES

1 Erika O. Almeida, Amilcar C. Freitas Júnior, Eduardo P. Rocha, Roberto S. Pessoa, Nikhil Gupta, Nick Tovar and Paulo G. Coelho (2012). Critical Aspects for Mechanical Simulation in Dental Implantology, Finite Element Analysis - From Biomedical Applications to Industrial Developments, Dr. David Moratal (Ed.), ISBN: 978-953-51-0474-2, InTech, DOI: 10.5772/38619.

2 Brozini T, Petridis H, Tzanas K, Garefis P. A meta-analysis of prosthodontic complication rates of implant supported fixed dental prostheses in edentulous patients after an observation period of at least 5 years. Int J Oral Maxillofac Implant. 2011; 26: 304-31.

3 Romeed SA, SL Fok, H.Wilson. A Comparison of 2D and 3D finite element analysis of a restored tooth. J Oral Rehab 2006; 33: 209-15.

4 Burak Ozcelik, T.E.Ersoy, B.Yilmaz. Biomechanical evaluation of tooth and implant-supported fixed dental prostheses with various non-rigid connector positions: a finite element analysis. J Prosthodont. 2011; 20: 16-28.

5 Freitas J, AC, EP Rocha, PH Santos et al. Mechanics of the maxillary central incisor. Influence of the periodontal ligament represented by beam elements, Computer Methods Biomechanical Biomedical Engineering, 2010; 13: 515-21.

6 Cahoon P, Hannam AG. Interactive modeling environment for craniofacial reconstruction. Visual data exploration and analysis. SPIE Proc. 1994: 206-15.

7 Sagat G, Yalcin S, Gultekin BA, Mijiritsky E. Influence of arch shape and implant position on stress distribution around implants supporting fixed full-arch prosthesis in edentulous maxilla. Implant Dent. 2010; 19: 498-508.

8 Bayraktar M. The influence of crown-implant ratio and dental implant parameters on implant and peri-implant bone: A finite element analysis. Istanbul University, Institute of Health Science, Department of Prosthetic Dentistry. PhD Thesis.

9 Chu CM, Huang H L, Hsu JT, Fuh LJ. Influences of internal tapered abutment designs on bone stresses around a dental implant: three-dimensional finite element method with statistical evaluation. J Periodontol. 2012; 83: 111-18.

10 Arisan V, Karabuda ZC, Avsever H, Özdemir T. Conventional Multi-Slice Computed Tomography (CT) and Cone-Beam CT (CBCT) for Computer-Assisted Implant Placement. Part I: Relationship of Radiographic Gray Density and Implant Stability. Clinical implant dentistry and related research. 2012.

11 Fajardo RJ, Ryan T, Kappelman J. Assessing the accuracy of high-resolution X-ray computed tomography of primate trabecular bone by comparisons with histological sections. Am J Phys Anthropol. 2002; 118: 1-10.

12 Magne P. Efficient 3D finite element analysis of dental restorative procedures using micro-CT data. Denl Mater. 2007; 23: 539-48.

13 Huang HL, Hsu JT, Fuh LJ, Tu MG, Ko CC, Shen YW. Bone stress and interfacial sliding analysis of implant designs on an immediately loaded maxillary implant: a non-linear finite element study. J Dent. 2008; 36: 409-17.

14 Okumura N, Stegaroiu R, Kitamura E, Kurokawa K, Nomura S. Influence of maxillary cortical bone thickness, implant design and implant diameter on stress around implants: a three-dimensional finite element analysis. J Prosthodont Res. 2010; 54: 133-42.

15 Geng JP, Tan KB, Liu GR. Application of finite element analysis in implant dentistry: a review of the literature. 2001; 85: 585-98.

16 Sevimay M, Turhan F, Kilicarslan M, Eskitascioglu G. Three-dimensional finite element analysis of the effect of different bone quality on stress distribution in an implant-supported crown. J Prosthet Dent. 2005; 93: 227-34.

17 Wakabayashi N, Ona M, Suzuki T, Igarashi Y. Nonlinear finite element analyses: advances and challenges in dental applications. J Dent. 2008; 36: 463-71.

18 Tada S, Stegaroiu R, Kitamura E, Miyakawa O, Kusakari H. Influence of implant design and bone quality on stress/strain distribution in bone around implants: a 3-dimensional finite element analysis. Int J Oral Maxillofac Implant. 2003; 18: 357-68.

19 Papavasiliou G, Kamposiora P, Bayne S, Felton D. 3D-FEA of osseointegration percentages and patterns on implant-bone interfacial stresses. J Dent. 1997; 25: 485-91.

20 Geng JP, Xu W, Tan K, Liu G. Finite element analysis of an osseointegrated stepped screw dental implant. J Oral Implant. 2004; 30: 223-33.

21 Davies JE. Understanding peri-implant endosseous healing. J Dent Educ. 2003; 67: 932-49.

22 Geng J, Yan W. Application of the finite element method in implant dentistry. Springer. 2008: 81-89.

23 Chou HY, Jagodnik JJ, Müftü S. Predictions of bone remodeling around dental implant systems. J. Biomech. 2008; 41: 1365-73.

24 Frost H. Bone “mass” and the “mechanostat”: a proposal. The anatomical record. 1987; 219: 1-9.

25 Cowin S, Nachlinger RR. Bone remodeling III: uniqueness and stability in adaptive elasticity theory. J Elast. 1978; 8: 285-95.

26 Saab XE, Griggs JA, PowersJM, Engelmeier RL. Effect of abutment angulation on the strain on the bone around an implant in the anterior maxilla: a finite element study. J Prosthet Dent. 2007; 97: 85-92.

27 Friberg B, Ekestubbe A, Sennerby L. Clinical outcome of Bra˚nemark System implants of various diameters: a retrospective study. Int J Oral Maxillofac Implant. 2002; 17: 671-77.

28 Ugural AC, Fenster SK. Advanced strength and applied elasticity, Pearson Education.

29 Teixeira E, Sato Y, Akagawa Y, Shindoi N. A comparative evaluation of mandibular finite element models with different lengths and elements for implant biomechanics. J Oral Rehabil. 1998; 25: 299-03.

30 Shunmugasamy VC, Gupta N, Pessoa RS, Janal MN, Coelho PG. Influence of clinically relevant factors on the immediate biomechanical surrounding for a series of dental implant designs. J.Biomech Eng. 2011; 133: 031005.

31 Chun HJ, Cheong SY, Han JH, Heo SJ, Chung JP, Rhyu IC, Choi YC, Baik HK, Ku Y, Kim MH. Evaluation of design parameters of osseointegrated dental implants using finite element analysis. 2002; 29: 565-74.

32 Olmsted MJ et al. Human bite force: the relation between EMG activity and bite force at a standardized gape. Am J Phys Anthropol. 2005: 160-61.

33 Rangert B, Jemt T, Jörneus L. Forces and moments on Branemark implants. Int J Oral Maxillofac Implant. 1989; 4: 241.

34 Mericske-Stern R, Piotti M, Sirtes G. 3-D in vivo force measurements on mandibular implants supporting overdentures. A comparative study. Clin Oral Implants Res. 1996; 7: 387-96.

35 Fontijn-Tekampl E, Slagter A, Van't Hof M, Geertman M, Kalk W. Bite forces with mandibular implant-retained overdentures. J Dent Res. 1998; 77: 1832-39.

36 Akyuz E, Braun JT, Brown NA, Bachus KN. Static versus dynamic loading in the mechanical modulation of vertebral growth. Spine. 2006; 31: 952-58.

37 Hasan I, Rahimi A, Keilig L, Brinkmann KT, Bourauel C. Computational simulation of internal bone remodelling around dental implants: a sensitivity analysis. Comput Methods Biomech Biomed Eng. 2012; 15: 807-14.

38 Pessoa RS, Muraru L, Júnior EM, Vaz LG, Sloten JV, Duyck J, Jaecques SV. Influence of Implant Connection Type on the Biomechanical Environment of Immediately Placed Implants–CT-Based Nonlinear, Three-Dimensional Finite Element Analysis. Clini Implant Dent Relat Res. 2010; 12: 219-34.

39 Meriç G, Erkmen E, Kurt A, Eser A, Oezden AU. Biomechanical comparison of two different collar structured implants supporting 3-unit fixed partial denture: A 3-D FEM study. Acta Odontologica Scandinavica. 2012; 70: 61-71.

40 Ciftci Y, Canay S. The effect of veneering materials on stress distribution in implant-supported fixed prosthetic restorations. Int J Oral Maxillofac Implant. 2000; 15: 571-582.

41 Cattaneo P, Dalstra M, Melsen B. The finite element method: a tool to study orthodontic tooth movement. J Dent Res. 2005; 84: 428-33.

42 Hsu ML, Chen FC, Kao HC, Cheng CK. Influence of off-axis loading of an anterior maxillary implant: a 3-dimensional finite element analysis. Int J Oral Maxillofac Implant. 2007; 22: 301-9.