Finite Element Analysis in Dentistry

K.Meena Anand, Syed Suhail, Vadlamudi Murali Krishna, Satheesha BC, Rajesh M, Meera A, Anand KM

***Abstract*— The constant pursuit of the scientific invention, R&D and adaption of cutting edge technology is an essential component of a world-view that changes the realities on how we live and survive today. The adaptation of new technology is growing tremendously in all fields including medicine and dentistry. It is getting more advanced day by day replacing all sort manual and tedious process in Dentistry creating a happy smile in a short time. Finte element analysis involves a series of computational procedures to calculate the stress in each element, which performs a model solution. Such a structural analysis allows the determination of stress resulting from external force, pressure, thermal change, and other factors. It is useful for indicating the mechanical aspects of biomaterials and human tissues which are difficult to be measured and analyzed in-vivo moreover it could consume a undefined timeliness. The results which are obtained can be studied using software where we can visualize within the FEM environment at variety of parameters and to identify the implications of finite element analysis. Analysis of stress of dental structures has been a topic of interest in the recent years with an objective of determining stresses in the dental structures and improvement of the mechanical strength of these structures. The present paper is on basic concept, applications and limitations of finite element method (FEM) in dentistry.**

***Index Terms*— Finite element analysis, Stress analysis, Enamel, bone, Periodontal ligament.**

I. Introduction

Finite element analysis is a powerful tool for numerical solution of a wide range of engineering problems. The application of finite element analysis is used for formation and stress analysis of automotive, aircraft, building and bridge structures, to stress analysis of heat flow, magnetic flux seeping and other problems. It was used for structural solving the structural mechanical problems. It was recognized as general procedure of numerical approximation to all physical problems that can be modeled by different equation description.

Analysis of stress of dental structures has been a topic of interest in the recent years with an objective of determining stresses in the dental structures and improvement of the mechanical strength of these structures. Stresses in dental structures have been studied by various techniques, such as brittle coating analysis, strain gauges, holography, 2-dimensional (2D) and 3-dimensional (3D) photo elasticity, finite element analysis (FEA), digital moiré interferometric investigation and other numerical methods. Most of the stress analysis of dental structures was carried out using the photoelastic technique. The advantages of using photoelastic study are that it can quantify stresses throughout a 3D-structure and determine stress gradients. However, it requires a bier fringent material and is more difficult with complex geometries1.

A more recent method of stress analysis, developed in 1956 in the aircraft industry was the finite element method (FEM). Initially, this technique was used widely only in aerospace engineering, slowly due to its flexibility of the method to model any complex geometries and provide instant results, it made its presence felt in dentistry. The Finite Element Method was first used in dentistry in the 1970's to replace photo elasticity tests2.

Finite element analysis is one of the popular numerical methods in stress analysis. This method involves a series of computational procedures to calculate the stress and strain in each element, by which it performs a model solution. Finite Element Method (FEM) and Finite Element Analysis (FEA) are one and the same. FEA is more popular in industries and FEM at universities. FEM circumvents / avoids many of the problems of material analysis by allowing, one to calculate physical measurements of stress within a structure3.

II. Concept of Finite Element Analysis:

There are 3 methods to solve any engineering problems namely, analytical method, numerical method, and experimental method4. The FEM is a numerical procedure used for analyzing structures and consists of a computer model of a material or design that is stressed and analyzed for specific results. FEM uses a complex system of points (nodes) and elements, which make a grid called as mesh.

This mesh is programmed to contain the material and structural properties (elastic modulus, Poisson's ratio, and yield strength), which defines how the structure will react to certain loading conditions. The mesh acts like a spider web, in that, from each node there extends a mesh element to each of the adjacent nodes. The main basic theme is to make calculations at only limited (finite) number of points and then interpolate the results for the entire surface or volume (domain).

Any continuous object has infinite degree of freedom (dofs) and it is not possible to solve the problem in this format. FEM reduces the dofs from infinite to finite with the help of meshing (nodes and elements) and all the calculations are

Manuscript received July 31, 2014.

K. Meena Anand, Department of Periodontology, MCODS, Manipal India, 9844373832

Syed Suhail, Director - Business Development at Sify Toronto Cannada. +16477066351.

Murali Krishna V, Department of Anesthesiology. Sunrise hospitals, Vijayawada, 9440978556.

Satheesha BC, Department of Internal Medicine, Subbiah Medical College, Shimoga. 7795488919.

Meera Avadhani, Department of Periodontology, MCODS, Manipal, 8050852469. 9739921546

Anand KM. Department of Microbiology, MMMC,Manipal. 9844009873.

Rajesh M. Department of Electrical & Electronics Engineering. MIT, Manipal.9739921546.

made at limited number of nodes5. Using these functions and the actual geometry of the element the equilibrium equations between the external forces acting on the elements and the displacements occurring on its nodes can be determined.

Theoretically, the shape and the size of elements determine the results. Finite element analysis started with triangular elements and these elements being stiffer, resulted in less stress and displacement. Later, quadrilateral elements were used for accuracy of results. Currently polyhedral mesh work is in research. Increasing the number of calculation points (nodes and elements) improves accuracy. Increasing the number of lines reduces error margin in finding out the area of a circle. The number of straight lines is equivalent to the number of elements in FEM.

In the industry, 2 types of analysis are generally used, 2-D modeling and 3-D modeling. 2-D modeling is comparatively simple and it allows the analysis to run on a relatively normal computer, but sometimes it tends to yield less accurate results. 3-D modeling produces more accurate results, but it can run only on the fastest computers effectively. For 2-D analysis, the elements are triangular or quadrilateral and in 3-D analysis 10, 12, or 14 faces are used. In some situations mass, spring, damper, and gap elements are also used.

FEM is performed with material properties that can be isotropic or anisotropic 6. All real-life materials are anisotropic, but it is simplified into isotropic properties or orthotropic properties (different properties along 3 axes, namely- x, y, and z). Elastic modulus, Poisson's ratio (strain in the lateral direction to that in the axial direction when an object is subjected to tensile loading)7, and yield strength for the materials are applied. The analysis is performed as linear static analysis or non-linear analysis depending on the allocation of appropriate physical characteristics to the different parts of the structure8.

Linear analyses are valid if the structure exhibits a linear stress–strain relationship up to a stress level known as the proportional limit, and all the volumes are bonded as one unit. However, the validity of a linear static analysis may be questionable when the study objectives are to explore more realistic situations that are usually encountered in the intraoral environment.

The application of the nonlinear FEM in dentistry seems to be interesting, for example in: nonlinear simulation of periodontal ligament properties, plastic and viscoelastic behaviors in materials, tooth-to-tooth contact analyses, analyses of contact in implant structures and interfacial stress in restorations. However, there are difficulties, for example: the dynamic behavior of the PDL is an aspect to be considered, and simulation using nonlinear analysis would be more realistic. Due to complex structure, the exact mechanical properties of PDL are poorly understood. Some simplifications and assumptions are common in FEA. These practices are allowed, but their impact on the conclusions should be carefully taken into account. The eventual result of any FEM is the normal and shearing stress values of the structure upon loading. The failure criteria are measured by Von-Mises stresses9.

III. STEPS IN USING FEA

Steps using in FEA are discretization of problem, imaging, meshing, boundary conditions, types of solutions

Discretization of problem

Solving a real life problem with continuous material approach is difficult. The basic of all numerical methods is to simplify the problem by discretizing (discontinuation) it. Nodes work like atoms with gap in between filled by an entity called as element10,11. Calculations are made at nodes and results are interpolated for elements.

There are two approaches to solve any problem. One is continuous approach (all real life components are continuous) and discrete approach (equivalent mathematical modeling). All the numerical methods including finite element follow discrete approach. Meshing (nodes and elements) is nothing but discretization of a continuous system with infinite degree of freedoms to finite degree of freedoms10.

Imaging:

Imaging and three-dimensional reconstruction: Innovations in computerized tomography (CT), magnetic resonance imaging (MRI), and confocal microscopy have revolutionized biological imaging. It is now possible to capture serial sections of any structure virtually and generate intricately detailed three dimensional reconstructions. Three dimensional surface reconstructions created from CT scans are used as templates for three-dimensional finite element models. Initial three-dimensional surface reconstructions are typically quite rough and require significant editing before they can be imported into a Finite Element tool and successfully meshed as a finite element model.

Image processing: Editing the three dimensional image are the most time intensive step in building Finite Element models of biological structures. The ultimate goal of three-dimensional image processing is to generate a "water-tight" surface model that can be imported into and successfully manipulated in Finite Element software.

The most important aspect of the simplification process of three-dimensional images involves smoothening and removing details in selected areas of the model. Three dimensional surface representations are composed of connected polygons and are often referred to as 'polygon models'. The more polygons a model contains, the greater is its reliability to the object it represents and larger is its size. Image processing is the most labor-intensive aspect of conducting Finite Element analyses of biological structures.

Meshing: FEM uses a complex system of points (nodes) and elements, which make a grid called as mesh. Basic theme of FEA is to make calculations at limited (finite) number of points and then interpolate the results for entire domain (surface or volume). Any continuous object has infinite degrees of freedom and it is just not possible to solve the problem in this format. FEA reduces degrees of freedom from infinite to finite with the help of discretization i.e. meshing (nodes and elements)10.

Two-dimensional and three dimensional meshing: Two dimensional modeling is comparatively simple and it allows the analysis to run on a relatively normal computer, sometimes it tends to yield less accurate results. Three dimensional modeling produces more accurate results, but it can run only on the fastest computers effectively. The element shapes for two dimensional analyses, are triangular, quadrilateral, and in three dimensional analysis element shapes are tetra, penta, hexa and pyramid.

Boundary conditions: Boundary condition is application of force and constraint. Different ways to apply force and moment are concentrated load (at a point or single node), force on line or edge, distributed load (force varying as equation), bending movements and torque. After fixing the boundary conditions the software is run for determining stresses & strains using linear static analysis & non-linear analysis10.

Types of solutions:

The analysis is done to assess the stresses acting upon the materials during function in the oral cavity by applying various material properties12, 13. These stresses are normal or principal stress which acts perpendicular to the cross section and causes elongation or compression and shear stress which acts parallel to the cross section and causes distortion (changes in original shape).

Whenever an elastic body is subjected to loads in its 3 dimensions, the stresses will get developed along the principal axis of the body. These are the principal stresses. There are three "principal stresses" that can be calculated at any point, acting in the x, y, and z directions. These stresses should not exceed the yield stress of the material. There is a convention on listing the three principal stresses which makes the 'first' one the maximum of the three, and the 'third' one the minimum, which can be the maximum compressive (negative) stress, but may actually be a positive stress.

Maximum principal stress: The maximum principal stress gives the value of stress that is normal to the plane in which the shear stress is zero. The maximum principal stress helps you understand the maximum tensile stress induced in the part due to the loading conditions.

Minimum principal stress: The minimum principal stress acts normally to the plane in which shear stress is zero. It helps you to understand the maximum compressive stress induced in the part due to loading conditions.

Von Mises stress: The von Mises criterion is a formula for calculating whether the stress combination at a given point will cause failure. The von Mises criterion is a formula for combining three principal stresses into an equivalent stress, which is then compared to the yield stress of the material. The yield stress is a known property of the material and is usually considered for the failure stress. If the "von Mises stress" exceeds the yield stress, then the material is considered to be at the failure condition. The von Mises theory is used for ductile materials such as metals and evaluates stresses in both static and dynamic conditions14.

IV. Software used for finite element analysis

The various software used in Finite Element Analysis are Abaqus Explicit, Ansys, Dytran, Femfat, Hypermesh , Ls - dyna , Madymo , Magmasoft, MSC Nastran, Pro mechanica, Star-CD, Tosca, Unigraphics, etc. 15, 16

V. Applications of finite element analysis

FEM technique is widely used in structural engineering. It helps to evaluate a detailed and complex structure in a computer, during the planning of the structure. It is also used to predict and estimate the damages in the electrical fields. The demonstration in the computer of the adequate strength of the structure and the possibility of improving the design during planning can justify the cost of this analysis work. FEA has also been known to increase the rating of structures that were significantly overdesigned and built many decades ago. It is also used in optimization of sheet metal blanking process. FEM is widely applied in commercial fields because of its incredible precision in obtaining results. Its applications have also increased because of decreasing cost and increasing technical up-gradation of the computer.

For complex structures, the simplifying assumptions required to make any calculations possible can lead to a conservative and heavy design. A considerable factor of ignorance can remain as to whether the structure will be adequate for all design loads. Significant changes in designs involve risk. Designs will require prototypes to be built and field tested. The field tests may involve expensive strain gauging to evaluate strength and deformation. With FEA, the weight of a design can be minimized, and there can be a reduction in the number of prototypes built. Field testing will be used to establish loading on structures, which can be used to do future design improvements via FEA17.

VI. Application of finite element analysis for dental use:

The use of finite element analysis in the field of oral research was done by Thresher in 19732 and Farah in 197318. Polson (1980)19 and Gher (1996)20 stated that finite element analysis provide information and insight into trauma associated response, repair, and accompanying adaptive response. However, they do not duplicate the dynamics of human masticatory function or rule out the significant difference between animals and man.

FEA has been applied for the description of form changes in biological structures in the area of growth and development21. To know physiological values of alveolar stresses which is important for the understanding the stress related bone remodeling and it also provides guideline for the design of dental implants22. The analysis of stresses produced in the periodontal ligament under different loading conditions21, 23. To study stress distribution in supporting structures of tooth in relation to different designs of fixed and removable prosthesis24, 25. It is used to enhance the design of dental restorations26. To investigate stress distribution in tooth with cavity preparation and biomechanical preparation during root canal treatment27.

VII. Enamel and finite element analysis:

Thresher R.W and Saito (1973) 2also did a stress analysis of a homogeneous and a non-homogeneous human tooth using a finite element method. They concluded that the enamel portion of the tooth carries the major portion of the load and the load is transferred into surrounding bone structure remarkably high on the tooth root.

A 3D Finite element analysis and strain gauge was used by by D.Palmara (2000)28 to investigate variations in strains in enamel under different pattern of occlusal loading in extracted teeth. The study concluded that magnitude, direction and character of strain in cervical region are dependent on pattern of loading.

Farah and Craig (1973)18 analyzed a restored axisymmetric first molar and compared different types of finish lines and their effect on the marginal configuration with the help of a finite element method. They found that the chamfer geometry exhibit exhibited the most uniform stress distribution.

Rees JS and Hammadeh M 200429 conducted a study to examine what effects the undermining of the buccal cervical enamel as a mechanism of abfraction lesion formation would have on stress distribution in upper teeth. 2d model of finite element meshes of upper incisor, canine, and first premolar and supporting periodontal ligament and alveolar bone was developed. The study concluded that undermining of buccal cervical alveo dental junction cause increase cervical stress profile which cause crack initiation in enamel leading to bulk loss.

VIII. Bone remodeling and finite element model:

Bone remodeling is always been an area of debate in FEM. Stress distribution in a model of FEA is not only dependent on loading configuration but also on geometry of the structure and property of the material (Huiskes and Chao 1983)30. In FEA alveolar bone is modeled as homogenous tissue approximating a regular geometric shape like cube cylinder etc. and the root shaped as cones or paraboloids (Vollmer et al 1999)31.

The model are isotropic, the actual bone is anisotropic and heterogeneous. Studies have been performed by simplifying the cancellous bone to a block and ignoring its trabecular structure. Stegaroui 199832, O'Mahony 200133, Sutpideler 200434 have reported that highest bone stress occur in the cortical bone around the neck of the implant. Analysis of cancellous bone stress / strain have found the highest stress / strain concentrated near the implant apex or near the interface with cortical bone depending on load direction and type of stress/ strain.

Finite element method was used to simulate the effect of the alveolar bone loss on orthodonticaly induced stress in periodontal ligament of maxillary 1st molar by Jeon PD, Turley PK, Ting K 200135. 3D models of tooth with different levels of bone height was constructed to estimate reduction in force and increase in moment to force (M/F) ratio to obtain evenly distributed stress in the periodontal ligament of tooth with horizontal bone loss. The study indicated that force reduction and increased M/F is required to achieve uniform stress in periodontal ligament of tooth with bone loss.

Quantification of pattern of stress changes produced in periodontal ligament with progression of alveolar bone loss by finite element method to stimulate secondary trauma from occlusion by Allahyar (2004). The study concluded that 2.5mm of alveolar bone loss can be considered as a limit beyond which stress alterations were accelerated. Based on FE analysis alveolar bone loss increases stress produced in the periodontal ligament inspite of applying same vector of force36.

IX. Periodontal ligament and finite element analysis:

The material properties of periodontal ligament had been modeled as linear elastic with elastic modulus ranging from 0.07 to 100MPa (Tanne et al 198737 Andersen at al 199138, Jones et al 2001)39 studies by Pietzak et al 200240, Poppe at al 200241 have modeled periodontal ligament as either a non-linear fiber reinforced material or with viscoelastic properties.

Reinhardt 1984 used 2 dimensional finite element analyses to calculate principal periodontal ligament stress in primary and secondary occlusal trauma. It was found that the greatest compressive stress was seen near the alveolar crest and in apical one half of the roots for all loads at all bone levels. Centric contact loads produced less ligament stress than protrusive contact. Reduction of alveolar bone height had little effect on degree of periodontal ligament stress until 6mm / 60% of bone support has been lost42.

The risk of root resorption was evaluated using the FEM of extracted max 1st premolar and simulating the distribution of hydrostatic pressure on pdl of these models. FEM of these teeth were created based on the computed tomography. (micro CT, voxel size 35microns). The study concluded that if hydroststic pressure exceeds human capillary blood pressure in the PDL, the risk of root resorption increases. If hydrostatic pressure rises above 0.0047 MPa, decisive condition for root resorption is fulfilled. Force controls more important factor to achieve optimal tooth mobility43.

Venturini (2009) studied to evaluate stress distribution in human periodontal ligament in masticatory, para-functional load and traumatic loads.by FE analysis. The study concluded that the tensile stress generation observed along the internal and external aspect of PDL, especially at cervical and middle third44.

Tanne in 1998 used finite element method to quantify the magnitude of tooth mobility in adolescent and adults and to investigate difference in biomechanical response of tooth and periodontium to orthodontic forces. The magnitude of tooth mobility was greater in adolescent than in adult group. As the stress level is determinant in inducing transient periodontal ligament hyalinization which would be causative factor for delayed tooth movement and pain during orthodontic treatment for adults45.

In 2013, Christof Holberg conducted Finite element analysis of mono and bicortical mini-implant Stability. Anisotropic Finite Element Method (FEM) models of the mandibular bone, including teeth, periodontal ligaments, orthodontic braces, and mini-implants of varying length, were created based on the Computed Tomography data. All mini-implants with varying insertion were typically loaded, and the induced effective stress was calculated in the cervical area of the cortical bone. They concluded that the deeper the mini-implant was anchored, the lower were the effective stress values in the cervical region of the cortical bone. Bicortical implant anchorage is biomechanically more favorable than mono cortical anchorage46.

X. Limitations of Finite Element Analysis

Finite elements methods are extremely versatile and powerful and can enable designers to obtain information about the behavior of complicated structures with almost arbitrary loading. In spite of the significant advances that have been made in developing finite element packages, the results obtained must be carefully examined before they can be used10.

The most significant limitation of FEA is that the accuracy of the obtained solution is usually a function of the mesh resolution.

Any regions of highly concentrated stress, such as around loading points and supports, must be carefully analyzed with the use of a sufficiently refined mesh. In addition, there are some problems which are inherently singular (the stresses are theoretically infinite). Special efforts must be made to analyze such problems10, 11.

The concern for any user is that, current packages can solve so many sophisticated problems, there is always a strong temptation to “solve” problems without doing the hard work of thinking through them and understanding the underlying mechanics and physical applications. Modern finite element packages are powerful tools that have become increasingly indispensable to mechanical design and analysis. However, they also make it easy for users to make big mistakes. Obtaining solutions with FEA often requires substantial amounts of computer and user time. Nevertheless, finite element packages have become increasingly indispensable to mechanical design and analysis.

XI. Conclusion:

FEM is an exceptionally a powerful tool in tackling many biomedical problems that are puzzling for conventional methods because of structural and material complexity. Modeling and simulation step used, saves time that is used for conducting the live experiment or clinical trial. this tool has been successfully employed in various areas of dentistry, It is important to substantiate the purpose of the work in to correctly apply. However this is an inception towards the modeling of more advanced building & restructuring of dentistry for a better world and happy tomorrow, which could last, hold and shine.

References

[1] Craig RG, Powers JM. Restorative dental materials. 11th ed. St. Louis, Missouri: Mosby, Inc; 2002. p. 110-1.

[2] Thresher RW, Saito GE. The stress analysis of human teeth. J Biomech 1973;6: 443-9.

[3] Wood I, Jawad Z, Paisley C, Brunton P. Noncarious cervical tooth surface loss: A literature review. J Dent 2008;36: 759-66.

[4] Tajima K, Chen KK, Takahashi N, Noda N, Nagamatsu Y, Kakigawa H.Three-dimensional finite element modeling from CT images of tooth and its validation. Dent Mater J 2009; 28: 219-26.

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

[6] Motta AB, Pereira LC, da Cunha AR. Finite element analysis in 2D and 3D models for sound and restored teeth. ABAQUS Users’ conference, 2006. p. 329-43.

[7] Anusavice KJ. Properties of dental materials. Phillip’s Science of Dental Materials. 10th ed. St. Louis Missouri: W.B Saunders Company; 1996. p. 58.

[8] Gallagher RH. Introduction. In: Gallagher RH. Finite element analysis: fundamentals. 4. ed. Englewood Cliffs: Prentice-Hall; 1975.cap. 1, p. 1-19.

[9] Kishen A, Ramamurty U, Asundi A. Experimental studies on the nature of property gradients in human dentine. J Biomed Mater Res 2000; 51:650-9.

[10] Nitin Gokhale, Sanjay Deshpande, Sanjeev Bedekar, Anand Thite. Practical finite element analysis. 1st ed. Finite to infinite; 2008.

[11] O. C. Zienkiewicz, R.L Taylor. Some preliminaries: the standard discrete system. In: The finite element method, Volume 1: The basics. 5th ed.Butterworth Heinemann; 2000.

[12] O’Brien WJ. Dental materials and their s e l e c t i o n . 2 n d e d . ch i c a g o : Quintessence; 2002. p. 347.

[13] Yang HS, Lang LA, Molina A, Felton DA. The effects of dowel design and load direction on dowel and core restorations. J Prosthet Dent. 2001; 85:558-67.

[14] J. H. Argyris. Energy theorems and structural analysis. Butterworth, 1960 (reprinted from Aircraft Eng: 1954-5.

[15] Giuseppe Pelosi (2007). "The finite element method, Part I: R. L. Courant: Historical Corner".

[16] Babuska, Ivo; Uday Banerjee, John E. Osborn (June 2004). "Generalized Finite Element Methods: Main Ideas, Results, and Perspective". International Journal of Computational Methods 1 (1): 67–103.

[17] Peter Budgell, Finite Element Analysis with ANSYS: Information and Tips, 1998 http://www3.sympatico.ca/peter_budge ll/FEA_intro.html

[18] Farah J.W, CRAIG R.G. "Element Stress Analysis of a Axisymmeric Molar" J Dent Res, 1973: 53:859-864

[19] Polson AM. Interrelationship of inflammation and tooth mobility (trauma) in pathogenesis of periodontal disease. J Clinical Periodontol 1980;7: 351-360

[20] Gher ME. Non surgical therapy: Dental occlusion. Ann Periodontol 1996;1: 567-580

[21] Takeshita S, Sasaki A, Tanne K, Publico AS, Moss ML. The nature of human craniofacial growth studied with finite element analytical approach. Clin Orthod Res 2001; 4: 148-60.

[22] Borchers L, Reichart P. Threedimensional stress distribution around a dental implant at different stages of interface development. J Dent Res 1983; 62(2): 155-9.

[23] Atma r am GH, Mo h amme d H. Estimation of physiologic stresses with a natural tooth considering fibrous PDL structure. J Dent Res 1981;60(5):873-7.

[24] Yang HS, Thompson VP. A twodimensional stress analysis comparing fixed prosthodontic approaches to the tilted molar abutment. Int J Prosthodont 1991; 4(5): 416-24.

[25] Awadalla HA, Azarbal M, Ismail YH, el-Ibiari W. Three - dimensional finite element stress analysis of a cantilever fixed partial denture. J Prosthet Dent 1992; 68(2): 243-8.

[26] Toparli M, Gokay M, Aksoy T. Analysis of a restored maxillary second premolar tooth by using three-dimensional finite element method. J Oral Rehabil 1999;26: 157-64

[27] Versluis A, Messer HH, Pintado MR. Changes in compaction stress distributions in roots resulting from canal preparation. Int Endod J 2006; 39: 931-9.

[28] Palamara D1, Palamara JE, Tyas MJ, Messer HH. Strain patterns in cervical enamel of teeth subjected to occlusal loading. Dent Mater. 2000 Nov;16(6):412-9.

[29] Rees JS, Hammadeh M. Undermining of enamel as a mechanism of abfraction lesion formation: a finite element study. Eur J Oral Sci. 2004 Aug;112(4):347-52.

[30] Huiskes R, Chao EY. A survey of finite element analysis in orthopedic biomechanics: in the first decade. J biomechanics 1983;16: 385-409

[31] J Vollmer, R. Mencl, andH.Mülle. Improved Laplacian Smoothing of Noisy Surface Meshes. EUROGRAPHICS Association and Backwell publishers 1999

[32] Stegaroiu R, Sato T, Kusakari H, Miyakawa O. Stress distribution in bone related to implant- supported prosthesis design. Int J Oral Maxillofac Implants 1998;13: 82-90

[33] O'Mahony AM, Williams JL, Spencer P. Anisotropic elasticity of cortical and cancellous bone in the posterior mandible increases peri-implant stress and strain under oblique loading. Clin Oral Implants Res 2001;12: 648-657.

[34] Sutpideler M, Eckert SE, Zobitz M. Finite element analysis of effect of prosthesis height, angle of force application and implant offset on supporting bone. Int J Oral Maxillofac Implants 2004;19: 819-825.

[35] Jeon PD1, Turley PK, Ting K. Three-dimensional finite element analysis of stress in the periodontal ligament of the maxillary first molar with simulated bone loss. Am J Orthod Dentofacial Orthop. 2001 May;119(5):498-504.

[36] Allahyar Geramy. Secondary trauma from occlusion: Three-dimensional analysis using finite element method. Quintesscence International 2004;vol 35(10):835-843

[37] Tanne K, Sakuda M, Burstone C J. Three-dimensional finite element analysis for stress in the periodontal tissue by orthodontic forces. American Journal of Orthodontics and Dentofacial Orthopedics 1987; 92: 499–505

[38] Andersen K L, Pedersen E H, Melsen B. Material parameters and stress profiles within the periodontal ligament. American Journal of Orthodontics and Dentofacial Orthopedics 1991; 99: 427–440

[39] Jones ML, Hickma J, Middleton J, Knox J, Volp C. A validated finite element method of study of orthodontic tooth movement in the human subject. J Orthood 2001;28: 29-38

[40] Pietrzak G, Curnier A, Botsis J, Scherrer S, Wiskott A, Belser U. A nonlinear elastic model of the periodontal ligament and its numerical calibration for the study of toth mobility. Comput Methods Biomech Biomed Engin 2002; 5: 91-100

[41] Poppe M, Bourauel C, Jäger A. Determination of the elasticity parameters of the human periodontal ligament and the location of the center of resistance of single-rooted teeth a study of autopsy specimens and their conversion into finite element models. Journal of Orofacial Orthopedics 200;63: 358–370

[42] Reinhardt RR, Pao YC, Krecji RF. Periodontal ligament stress in the initiation of occlusal traumatism. J Periodont Res 1984;19: 238-246

[43] Hohmann A1, Wolfram U, Geiger M, Boryor A, Kober C, Sander C, Sander FGCorrespondences of hydrostatic pressure in periodontal ligament with regions of root resorption: a clinical and a finite element study of the same human teeth. Comput Methods Programs Biomed. 2009 Feb;93(2):155-61

[44] Isis A Venturini P Poiate, Adalberto Bastos de Vasconcellos, Ronaldo Barcellos de Santana, Edgard Poiate Three-dimensional stress distribution in the human periodontal ligament in masticatory, parafunctional, and trauma loads: finite element analysis. Journal of Periodontology 2009; 80(11):1859-67.

[45] Tanne K, Yoshida S, Kawata T, Sasaki A, Knox J, Jones ML. An evaluation of the biomechanical response of the tooth and periodontium to orthodontic forces in adolescent and adult subjects. Br J Orthod. 1998 May;25(2):109-15.

[46] Christof Holberg, Philipp Winterhalder, Ingrid Rudzki-Janson and Andrea Wichelhaus. Finite element analysis of mono- and bicortical mini-implant Stability. European Journal of Orthodontics 2013; 4(18): 1-7