Available online at http://www.journalcra.com

International Journal of Current Research
Vol. 9, Issue, 02, pp.46755-46760, February, 2017

INTERNATIONAL JOURNAL OF CURRENT RESEARCH

ISSN: 0975-833X

REVIEW ARTICLE

NEW VISTAS IN DENTISTRY : FEM A COMPLETE REVIEW

Dr. Ankita Jaiswal, Dr. Ragni Tandon, Dr. Kamlesh Singh, Dr. Aftab Azam and *Dr. Abhimanyu Rohmetra

Department of Orthodontics and Dentofacial Orthopaedics, Saraswati Dental College, Lucknow

ARTICLE INFO

Article History:
Received 16th November, 2016
Received in revised form
18th December, 2016
Accepted 02nd January, 2017
Published online 28th February, 2017

Key words:

Finite element analysis,
3 Dimensional, Orthodontics.

ABSTRACT

In recent years, stress analysis of dental structures has been a topic of interest with an objective of determining stresses in the dental structures and improvement of the mechanical strength of these structures. The study of orthodontic biomechanics requires the understanding of nature of stress and strain induced by orthodontic forces. FEM is an engineering method of calculating stresses and strains in all materials including living tissues. The finite element analysis provides the orthodontist with quantitative data that can extend the understanding of physiologic reactions that occur within the dentoalveolar complex. Such a structural analysis allows the determination of stress resulting from external force, pressure, thermal change, and other factors. The scope of the review covers various steps of finite element analysis, its applications in context to orthodontics and general dentistry.

Citation: Dr. Ankita Jaiswal, Dr. Ragni Tandon, Dr. Kamlesh Singh, Dr. Aftab Azam and Dr. Abhimanyu Rohmetra, 2017. "New vistas in dentistry : Fem a complete review", *International Journal of Current Research*, 09, (02), 46755-46760.

INTRODUCTION

The finite element method (FEM) is a numerical procedure for analyzing structures. Finite Element Analysis (FEA) has been widely used through numerical analysis that has been applied successfully in a large number of engineering and bioengineering areas since the 1960s. Usually Many people get confused between the terminologies FEM and FEA. In reality, both are one and the same. FEM is more popular at universities and FEA in industries. The most biomechanical research of the oral environment such as in orthodontics, implantology, restorative dentistry, endodontics, prosthodontics etc. have been performed in vitro. Such numerical techniques may yield an improved understanding of the reactions and interactions of individual tissues (Geng *et al.*, 2001; Kazuo Tanne *et al.*, 1987). The finite element analysis provides the orthodontist with quantitative data that can extend the understanding of physiologic reactions that occur within the dentoalveolar complex and, also been proven to be a precise and applicable tool for evaluating stress patterns in the implant system and the surrounding bone (Holmgren *et al.*, 1998).

History

In 1943, R. Courant developed this technique (Yettram *et al.*, 1972). He used Ritz method to minimize the various calculative procedures to gain an absolute solution to bio-mechanical system. In 1956, Turner *et al.* describe this method by developing a broader definition of this numeric analyses[5]. In 1976, Weinstein used this technique in implant dentistry to evaluate various loads of occlusion on the implant and adjacent bone. Since then, it is used in micro-computer as well as analysis of large-scale structural system (Yijunliu, 2003).

Advantages of FEM

- FEM is a non-invasive technique.
- It does not require extensive instrumentation.
- It can be applicable to linear and non-linear as well as solid and fluid-structural interactions.
- The study can be repeated as many times as the operator wants.
- Any problems can be split into a smaller number of Problems.
- By using FEA, It's very easy to simulate any biological condition in intra, pre and post-operative stages to achieve accurate and reliable results.
- Reproducibility does not affect the physical properties involved.
- Static and dynamic analysis can be done.
- This technique is very less time consuming so that the complicated studies which would take a very long duration to finish can now be evaluated in a lesser time frame.

****Corresponding author:* Dr. Abhimanyu Rohmetra,**
Department of Orthodontics and Dentofacial Orthopaedics, Saraswati Dental College, Lucknow

- Difference between 2D and 3D models (Poiate *et al.*, 2011) Table I

Table I. Difference between 2D and 3D techniques

2D	3D
Time and cost more effective method	Greater computational cost
Simpler, easier to build and less time consuming compared to the 3D model	Require a mesh refinement, more complex analysis and full assessments which yield accurate results
They do not represent the complexity of the real problem	Better visualization of internal areas

Shortcomings of FEM

- Inaccurate information, data, and interpretation will yield completely misguiding results.
- The progress in the technique will be limited until better defined physical properties for enamel, periodontal ligament, dentin and cortical and cancellous bone are available.
- Modeling human structures is very difficult because of their complex anatomy and lack of knowledge about their complete mechanical behaviors. Certain assumptions are bound to be accepted. Hence, results will depend on the personnel involved in the process.
- The tooth is treated as it is pinned to the supporting bone, which is considered to be rigid whereas the nodes connecting the tooth to the bone are considered fixed. This assumption will introduce some error.

Methodology in finite element analysis

The object to be analyzed is graphically simulated in a computer in the form of a mesh, which defines the geometry of the body being studied. This mesh is divided by a process known as discretization, further, into a number of subunits termed elements. These are connected to a finite number of points known as nodes.

Basic steps involved in carrying out FEA are:

A. Pre-processing.
B. Conversion of geometric model into finite element model.
C. Assembly/Material Property data representation.
D. Defining the boundary conditions.
E. Loading Configuration.
F. Processing.
G. Post-processing.

A. Pre-processing:

Construction of the Geometric model

The purpose of the phase is to represent geometry in terms of volume, areas, lines and points. Smooth or complicated objects can be represented by geometrically simple pieces that are (Elements) (Holmgren *et al.*, 1998). This can be achieved by 3D – CT scanner. Usually done for modeling living tissues or complex structures. For example maxilla, mandible or craniofacial skeleton 3D – Laser scanner. Usually done for modeling inanimate objects for example modeling of brackets.

B. Conversion of the Geometric model to Finite Element Model

The process of the dividing problem into several small elements which are connected with nodes is called Discretization. All nodes and elements must be numbered so that a setup of matrix connectivity is established. This affects the computing time. The elements could be one, two or three-dimensional and in different shapes. It is essential that the elements are not overlapping and are connected only at the key points, which are termed *nodes*. The joining of elements at the nodes and eliminating duplicate nodes is termed as '*Meshing*'. (Figure 1)

C. Material / Assembly Property data representation

Equations are developed for every element in the FEM mesh and assembled into a set of global equations which model the properties of the whole system. Poisons ratio and young's modulus are the properties which are minimally required.

D. Defining the Boundary Conditions

Suppose an element is constructed on the computer and a force is applied to it, it will behave like a free-floating rigid body and will undergo a rotatory or translatory motion or a combination of the two without undergoing deformation. To study the deformation, some degrees of freedom must be restricted (movement of the node in each direction x, y, and z) for some nodes. These such constraints are termed boundary conditions.

E. Loading configuration

Force application at various points of geometry and its configuration.

F. Processing

Solve the system of linear algebraic equation. The stresses are determined by the Hooke's law. Strains are procured from the displacement functions within the element Combined with Hooke's law.

G. Post-Processing

The output result from the Finite Element Analysis is primarily in the *numerical form*. It usually consists of nodal values of the field variables and its derivatives. For example in mechanical problems, the output is element stresses and nodal displacement. *Graphic outputs* and displays are usually more informative. The curves and contours of the field variable can be plotted and displayed. Also irregular shapes can be displayed and superimposed on unreformed shapes. The output is primarily in the form of color-coded maps. The quantitative analysis is determined by interpreting these maps.

Applications of FEM

- Mechanical / Aerospace/Civil/Automobile
- Engineering.
- Structure analysis (Static/dynamic, linear/no
- Linear.
- Thermal/fluid flows.
- Geomechanics.
- Electromagnetic.

- Orthopedic research
- Biomechanics.
- To investigate soft-tissue and skeletal responses to

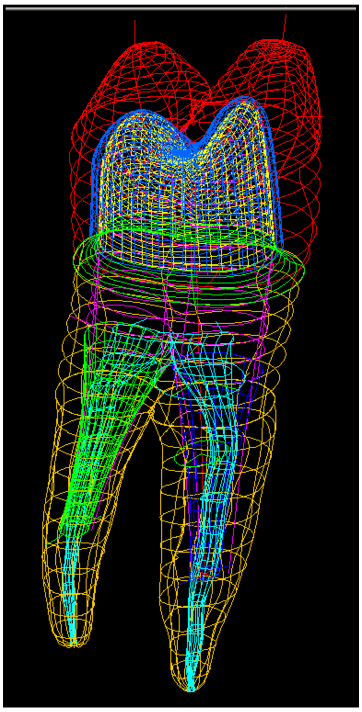

Figure 1.

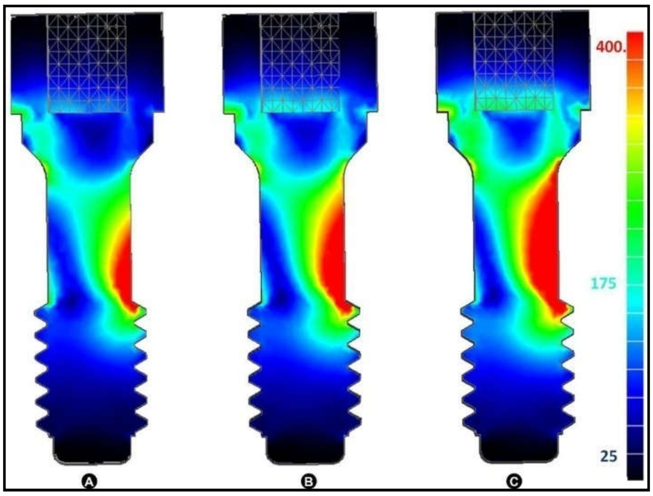

Figure 2.

Figure 3.

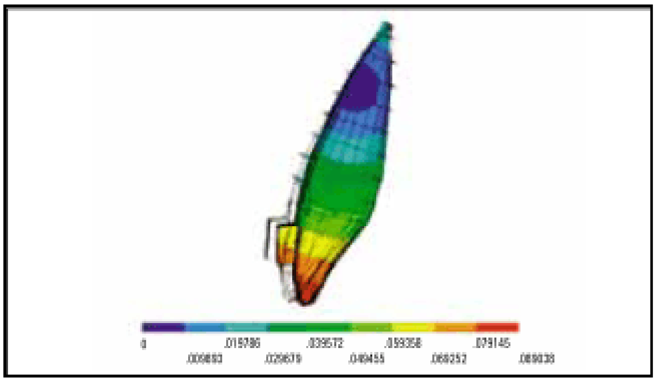

Figure 4.

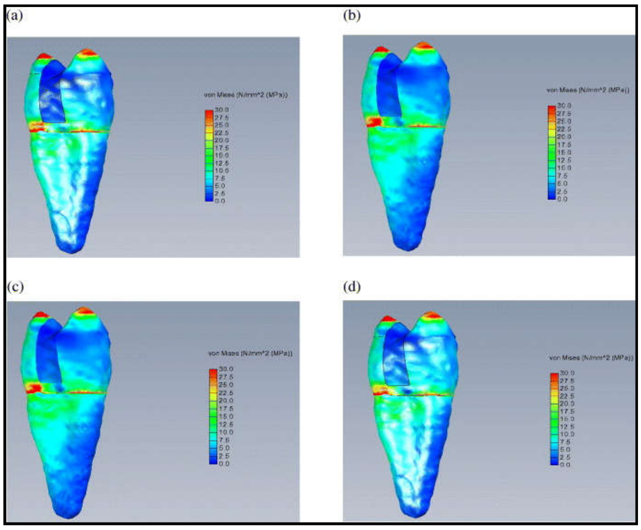

Figure 5.

Application of the finite element method in Dentistry

Implantology

Genj *et al.* (2001) reviewed the current status of FEA applications in implant dentistry and discussed findings from FEA studies. to achieve more realistic models, advanced digital imaging techniques can be used to generate model bone geometry in greater detail; the anisotropic and non-homogenous nature of the material needs to be considered, and boundary conditions must be refined. In addition, 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. (Figure 2) In a finite element study on immediately loaded implants, Ding *et al.* (2009) showed that the simulated masticatory force was better dissipated and the stress and strain around the implant neck were decreased when the implant diameter was increased. Several studies (Lozada *et al.*, 1994; Meijer *et al.*, 1996; Meijer *et al.*, 1992) using FEA have found that a higher risk of bone resorption occurs in the neck region of the implant. By using the FEM, the authors could compare the deformation and elastic modulus of different types of bone, which helps clinicians to understand the process of bone remodeling, for further improvements of their surgery techniques. Erslan *et al.* (2010) found that different implant thread forms did not affect the von Mises stress distributions in supporting bone structure, but produced different compressive stress intensities in the bone. Materials used in the study were assumed as homogenous and isotropic.

The elastic properties of the materials ([E] and [μ]) were determined from the literature. Chun *et al*. (2002) found that the square thread shape filleted with a small radius was more effective on stress distribution than other dental implants used in the analyses. Dos Santos *et al*. (2011) showed that the simulations with non-submerged implants showed higher values of stress concentration than those that were submerged. It was also demonstrated that soft liner materials presented better results than when the denture base was not relined. The height of the healing caps seems to have a direct influence on the stress distribution in the peri-implant bone during the healing period.

Siegele and Soltesz (1989) did a 3D FEM study to evaluate stress generation within the jawbone after insertion of implants of various shapes. They concluded that different implant shapes create different stress distributions in the jawbone. They also stated that conical implant implies distinctly higher stresses than cylindrical and screw-shaped implants. Himmlova *et al*. (2004) undertake a 3D FEM study to evaluate stress values produced at the implant-bone interface. They have taken implants of various lengths and diameter. They found Maximum stress areas at the neck of the implant. The maximum decrease in stress was found with increasing the diameter of implant up to 4.5 mm.

Orthodontics and Dentofacial Orthopaedics

FEM has proved to be a valid and reliable technique for evaluating the deformation and loading characteristics of complex structures following the application of orthodontic forces. Liang *et al*. (2009) generated 3-D finite element models of the maxilla and maxillary incisors to evaluate the torque control of maxillary incisors during retraction in lingual and labial orthodontic technique. 3D force system including intrusion, retraction and moment of the force was applied to predetermined points to simulate retraction in both the techniques. Distribution of strain and stress on PDL and adjacent bone as well as resultant movements of incisors were evaluated and compared. Results showed the same amount of forces results in uncontrolled tipping in labial and lingual and bodily movement techniques respectively. (Figure 3) Cattaneo *et al*. (2009) indicated that, following the application of an orthodontics loading regime, the concept of resorption is caused by compression, and formation is caused by tension. Furthermore, the same authors (2005) found that tension in the alveolar bone was far more predominant than compression. Sung *et al*. (2003) generated 3D finite element models of the mandibular incisors and mandible to evaluate the anti-tip and anti-rotation effect of the reverse curve of spee in lingual and labial orthodontics. Retraction force of 150 gm was applied to a canine in both the techniques. Every time amount of spee was increased from 0 mm to 4 mm. Resultant strain and stress on PDL were observed and compared. The comparison depicts, reverse curve of spee works well with the labial technique in comparison to lingual technique. Jones *et al*. (2001) validated a FEM model and discovered that the PDL is the main mediator of orthodontic tooth movement, revealing that PDL demonstrated an initial elastic response followed by a visco-elastic phase when subjected to a continuous load and the materials properties of periodontal ligament were difficult to quantify. Tanne *et al*. (1995) done a study to investigate the location of the center of resistance (CRe) in the nasomaxillary complex by the finite element analysis. A three-dimensional finite element model of the craniofacial complex was used. 9.8 N of force was applied in inferior and anterior direction at predetermined points which were perpendicular and parllel to the occlusal plane. After force application, 3D displacement of various anatomic points on maxillary dentition as well as nasomaxillary complex was observed. When the load was applied passing from the superior ridge of the pterygomaxillary fissure, the resultant movement was deduced to be a translation. This suggested that CRe of the nasomaxillary complex is situated on the posterosuperior ridge of the pterygomaxillary fissure, registered on the median sagittal plane. Qian *et al*. (2009) conducted a study by means of a combined numerical and experimental approach, to investigate the full-field distributions of displacement, strain and stress, and their generation with loading in the fresh periodontium under force applied externally. They concluded that the non-linear and time-dependent viscoelasticity of the PDL enables the acquisition of a full picture of detailed, realistic strain/stress fields, and deformation patterns of the entire fresh periodontium.

Clarice *et al*. (2009) done a study to evaluate level of stress in multi and single tooth system using 3D FEM technique. Two different models were made which consisted single mandibular incisor, mandibular canine and premolars respectively. Loads were applied on both models to simulate tipping orthodontic movement. Stress distribution observed stated that there was an elevated distortion strain at the crest of the alveolar ridge, while the compressive and tensile stresses coincide at apex of tooth resulting in root resorption. Multi-tooth system consisted increased stress levels in comparison to single tooth system. (Figure 4) Rudolph *et al*. (2001) undertook a study to evaluate stress generation at root apex during orthodontic tooth movements. A 3 D finite element model of a maxillary central incisor, periodontal ligament (PDL) and alveolar bone was constructed on the basis of anatomic morphology. The material properties of enamel, PDL, dentin and bone were compared for various possible tooth movements. Results showed that higher stresses were found concentrated at apex during intrusive, extrusive, and rotational movements.

Operative Dentistry and Endodontics

FEM has been used to analyze stresses generated in teeth as well as restorations. It has proven to be a useful tool for understanding tooth biomechanics and the biometric approach in restorative dentistry. Asmussen *et al*. (2008) studied Class I and Class II resin composite restorations and the influence of the modulus of elasticity of stresses generated by occlusal loading. They concluded that in order to reduce the risk of marginal deterioration, resin composite occlusal restorations should have a high modulus of elasticity. They also stated that the elastic constants of materials and the restorations were loaded centrally with a force of 100 N in the axial while Yamamoto *et al*. (2007) determined elastic modulus and Poisson’s ratio by nano indentation and a total occlusal load of 600 N were applied along the cusp. (Figure 5) Ichim *et al*. (2007) did a 3D FEM study to investigate co-relation of cavity shape, depth and occlusal forces with the durability of GIC restoration. They also concluded that shape and depth have no significant effect on restoration. They advised to re-adjust the interocclusal contacts for better retention of restoration. They also suggested occlusal re-adjustment of tooth contacts. Another study (Ichim *et al*., 2007) by the same authors using the nonlinear technique for crack propagation showed the mechanical failure of biomaterials in clinical loading conditions. Advanced studies by same authors on the elastic modulus of materials stated that more flexible materials with elastic modulus of 1 GPa should be used for cervical restorations for the better results (Ichim *et al*., 2007).

Magne *et al*. (1992) evaluated five models: natural tooth, MO and MOD cavities, MO and MOD endodontic access preparations and showed that the progressive tooth structure loss (MO to MOD to endodontic access) results into loss of cuspal stiffness. At different restorative steps, the cuspal widening was measured and correlated with existing experimental data for model validation and optimization. Campos *et al*. (2011) and Belli *et al*. (2005) investigated the use of ceramics and polymers in indirect restorations. They stated ceramic to be a preferable material for restoring because its structure keeps the stress inside and, therefore, transfers less stress through the tooth structure. Geometry, mechanical properties, and thickness of the restorative material can directly influence the load distribution in a tooth/restoration complex and consequently the results. Xavier *et al*. (2013) considered 3-D models as more reliable than the 2D models for analyzing the shear and micro shear bond strength tests. Subramaniam *et al*. (2007) did a 3D FEM study to analyze bending and torsional stresses in simulated models of nickel-titanium rotary instruments by Protaper and Profile. They stated that when loads are equal, the Protaper model showed a uniform distribution of stress and less elasticity when compared with Profile model. Few shortcomings of their study were that they ignored the variations in the taper of the Protaper and Profile instruments and also nonlinear mechanical behavior of the Ni-Ti material was not considered. Hong *et al*. (2003) undertook a 3D FEM study to find out a better method of condensation in the root canal to prevent vertical root fracture. They found that high stress was generate in vertical condensation technique on root canal walls than lateral condensation. But over forced lateral condensation with improper technique can be a reason for vertical root fracture (2016).

Conclusion

FEM has proven itself an extremely powerful tool in addressing many biomedical problems that are challenging for more conventional methods because of structural and material complexity. The modeling and simulation steps save time and money for conducting the live experiment or clinical trial. Although FEA is an accurate tool in assessing stress distribution, it is effective only for a given situation or set of values. FEA still needs laboratory validation to prove its results. Therefore, this tool has been successfully employed in various areas of dentistry, but it is extremely important to verify what the purpose of the study is in order to correctly apply FEM. More studies are required to use appropriate models.

REFERENCES

Asmussen E, Peutzfeldt A. 2008. Class I and Class II restorations of resin composite: an FE analysis of the influence of modulus of elasticity on stresses generated by occlusal loading. *Dent Mater*. 24(5):600-5.

Belli S, Eskitascioglu G, Eraslan O, Senawongse P, Tagami J. 2005. Effect of hybrid layer on stress distribution in a premolar tooth restored with composite or ceramic inlay: an FEM study. *J Biomed Mater Res Part B: Appl Biomater*. 74(2):665-8.

Campos RE, Soares CJ, Quagliatto PS, Soares PV, Oliveira Junior OB, Santos-Filho PCF *et al*. 2011. In vitro study of fracture load and fracture pattern of ceramic crowns: a finite element and fractography analysis. *J Prosthodont*, 20(6):447-55.

Cattaneo PM, Dalstra M, Melsen B. 2005. The finite element method: a tool to study orthodontic tooth movement. *J Dent Res*., 84(5):428-33.

Cattaneo PM, Dalstra M, Melsen B. 2009. Strains in periodontal ligament and alveolar bone associated with orthodontic tooth movement analyzed by finite element. *Orthod Craniofac Res*., 12(2):120-8.

Chun HJ, Cheong SY, Han JH, Heo SJ, Chung JP, Rhyu IC *et al*. 2002. Evaluation of design parameters of osseointegrated dental implants using finite element analysis. *J Oral Rehabil*., 29:565-74.

Clarice Field, Ionut Ichim, Michael V Swain, Eugene Chan, M Ali Darendeliler, *et al*. 2009. Mechanical responses to orthodontic loading: a 3-dimensional finite element multi-tooth model. *Am J Orthod Dentofacial Orthop*., 135:174-181.

Ding X, Zhu XH, Liao SH, Zhang XH, Chen H. 2009. Implant–bone interface stress distribution in immediately loaded implants of different diameters: a three-dimensional finite element analysis. *J Prosthodont*., 18(5):393-402.

Dos Santos MBF, Silva Neto JP da, Consani RLX, Mesquita MF. 2011. Three-dimensional finite element analysis of stress distribution in peri-implant bone with relined dentures and different heights of healing caps. *J Oral Rehabil*., 38(9):691-6.

Eraslan O. and Inan O. 2010. The effect of thread design on stress distribution in a solid screw implant: a 3D finite element analysis. *Clin Oral Invest*., 14(4):411-6.

Geng JP, Tan KB, Liu GR 2001. Application of finite element analysis in implant dentistry: a review of the literature. *J Prosthet Dent,* 85: 585-598.

Himmlová L, Dostálová T, Kácovský A, Konvicková S 2004. Influence of implant length and diameter on stress distribution: a finite element analysis. *J Prosthet Dent*, 91: 20-25.

Holmgren EP, Seckinger RJ, Kilgren LM, Mante F. 1998. Evaluating parameter of osseointegrated dental implant using finite element analysis – A two dimensional comparative study examining the effects of implants diameter, implant shape & load direction. *J Oral Implantology*, Vol.24 (2):80-88.

Hong J, Xia WW, Xiong HG 2003. Analysis of the effect on the stress of root canal wall by vertical and lateral condensation procedures. Shanghai Kou Qiang Yi Xue 12:359-361.

Ichim I, Li Q, Loughran J, Swain MV, Kieser J. 2007. Restoration of non-carious cervical lesions Part I. Modelling of restorative fracture. *Dent Mater*, 23: 1553-1561.

Ichim I, Schmidlin PR, Kieser JA, Swain MV. 2007. Mechanical evaluation of cervical glass-ionomer restorations: 3D finite element study. J Dent 35: 28-35.

Ichim IP, Schmidlin PR, Li Q, Kieser JA, Swain MV. 2007. Restoration of noncarious cervical lesions Part II. Restorative material selection to minimise fracture. *Dent Mater,* 23: 1562-1569.

Jones ML, Hickman J, Middleton J, Volp C. 2001. A validated finite element method study of orthodontic tooth movement in the human subject. *J Orthod*., 28(1):29-38.

Kazuo Tanne, Mamoru Sakuda, Charles J, 1987. Three-dimensional finite analysed for stress distribution in the periodontal tissue by orthodontic forces. *Am J Orthod Dentofacial Orthop.,* 92: 499-505.

Kulshrestha R, Tandon R, Kinger S, Rohmetra A, Singh RV. 2016. Obstructive sleep apnea in orthodontics: An overview. *Int J Orthod Rehabil*., 7:115-8.

Liang W, Rong Q, Lin J, Xu B 2009. Torque control of the maxillary incisors in lingual and labial orthodontics: a 3-dimensional finite element analysis. *Am J Orthod Dentofacial Orthop*., 135: 316-322.

Lozada JL, Abbate MF, Pizzarello FA, James RA. 1994. Comparative three-dimensional analysis of two finite-element endosseous implant designs. *J Oral Implantol*., 20(4):315-21.

Magne P. 2007. Efficient 3D finite element analysis of dental restorative procedures using micro-CT data. *Dent Mater,* 23: 539-548.

Mailath G, Stoiber B, Watzek G, Matejka M 1989. [Bone resorption at the entry of osseointegrated implants--a biomechanical phenomenon. Finite element study]. *Z Stomatol*, 86: 207-216.

Meijer HJ, Kuiper JH, Starmans FJM, Bosman F. 1992. Stress distribution around dental implants: influence of superstructure, length of implants, and height of mandible. *J Prosthet Dent*, 68(1):96-102.

Meijer HJ, Kuiper JH, Starmans FJM, Bosman F. 1992. Stress distribution around dental implants: influence of superstructure, length of implants, and height of mandible. *J Prosthet Dent*, 68(1):96-102.

Meijer HJ, Starmans FJM, Steen WHA, Bosman F. 1996. Loading conditions of endosseous implants in an edentulous human mandible: a three-dimensional, finite-element study. *J Oral Rehabil*., 23(11):757-63.

Poiate IA, Vasconcellos AB, Mori M, Poiate Jr. E. 2011. 2D and 3D finite element analysis of central incisor generated by computerized tomography. *Comput Methods Programs Biomed*, 104:292-9.

Qian L, Todo M, Morita Y, Matsushita Y, Koyano K. 2009. Deformation analysis of the periodontium considering the viscoelasticity of the periodontal ligament. *Dent Mater*, 25(10):1285-92.

Rudolph DJ, Willes PMG, Sameshima GT. 2001. A finite element model of apical force distribution from orthodontic tooth movement. *Angle Orthod*., 71: 127-131.

Siegele D. and Soltesz U 1989. Numerical investigations of the influence of implant shape on stress distribution in the jaw bone. *Int J Oral Maxillofac Implants,* 4: 333-340.

Subramanium V, Indira R, Srinivasan MR, Shankar P. 2007. Stress distribution in rotary nickel titanium instruments: A finite element analysis. *J Conserv Dent*, 10:112-118.

Sung SJ, Baik HS, Moon YS, Yu HS, Cho YS 2003. A comparative evaluation of different compensating curves in the lingual and labial techniques using 3D FEM. *Am J Orthod Dentofacial Orthop*., 123: 441-450.

Tanne K, Matsubara S, Sakuda M 1995. Location of the centre of resistance for the nasomaxillary complex studied in a three-dimensional finite element model. *Br J Orthod*., 22: 227-232.

Turner MJ, Clough RW, MactinandL HC 1956. Stiffness and defection analysis of complex structure. *J Aeronaut Science,* 23:805-823.

Xavier TA. and Ballester RY. 2013. A comparison between the capacity of 2D and 3D finite element models in analyzing the stress distribution in shear and microshear bond strength tests. *J Res Dent*, 1(1):41-54.

Yamamoto T, Takeishi S, Momo Y. 2007. Finite element stress analysis of indirect restorations prepared in cavity bases. *Dent Mater J*., 26(2):274-9.

Yettram AL, Wright KW, Pickard HM. 1972. Finite element stress analysis of crowns of normal and restored teeth. *Acta Orthop Scand.,* 3:304-4.

Yijunliu 2003. Introduction to FEM –CAE research laboratory university of Cincinnati U.S.A. Accessed on 24 January 2017 Available from URL http://ingenieria.udea.edu.co/~ingcivil/dinamica_de_estructuras/Apoyo/FEM_Lecture_Notes1.pdf

****