REVIEW ARTICLE

Finite element method: a lantern illuminating new horizons in treatment planning and research

Anil Sharma[1],Prakash V S[2], A K Chauhan[1]

Abstract

FEA is a computer-based process used for modeling complex products and systems. These simulations occur in a virtual environment for the purpose of 'solving' or finding a series of solutions to potentially complex performance issues. It is useful for all degrees of science. Specifically, as applied to material sciences and engineering, it is used to calculate the strength and behavioral characteristics of a material under conditions such as stress, vibration and deflection

Key words: *FEM, elements, nodes*

Introduction

Biomechanics is fundamental to any dental practice, including dental restorations, movement of misaligned teeth, implant design, dental trauma, surgical removal of impacted teeth, and craniofacial growth modification. Following functional load, stresses and strains are created inside the biological structures. Stress at any point in the construction is critical and governs failure of the prostheses, remodeling of bone, and type of tooth movement.

However, *in vivo* methods that directly measure internal stresses without altering the tissues do not currently exist. The advances in computer modeling techniques provide another option to realistically estimate stress distribution. Finite element analysis (FEA), a computer simulation technique, was introduced in the 1950s using the mathematical matrix analysis of structures to continuum bodies (Zienkiewicz and Kelly 1982).[1]

1. *Professor*
2. *Post graduate student*

Department of Orthodontics and Dentofacial Orthopedics

Correspondence Address

DrPrakash V S (P.G. student)

Kothiwaldental college and research centre, Moradabad.

prakash.vscaria@gmail.com

Over the past 30 years, FEA has become widely used to predict the biomechanical performance of various medical devices and biological tissues due to the ease of assessing irregular-shaped objectscomposed of several different materials with mixed boundary conditions. Unlike other methods (e.g., strain gauge) which are limited to points on the surface, the finite element method (FEM) can quantify stresses and displacement throughout the anatomy of a three dimensional structure. The FEM is a numerical approximation to solve partial differential equations (PDE) and integral equations (Hughes 1987, Segerlind 1984)[2] that are formulated to describe physics of complex structures (like teeth and jaw joints)

HISTORY

While it is difficult to quote a date of the invention of the finite element method, the method originated from the need to solve complex elasticity and structural analysis problems in civil and aeronautical engineering. Its development can be traced back to the work by A. Hrennikoff and R. Courant in the early 1940s.

In the USSR, the introduction of the practical application of the method is usually connected with name of Leonard Oganesyan. In China, in the later 1950s and early 1960s, based on the computations of dam constructions, K. Feng proposed a systematic numerical method for solving partial differential equations. The method was called the finite difference method based on variation principle, which was another independent invention of the finite element method. Although the approaches used by these pioneers are different, they share one essential characteristic: mesh discretization of a continuous domain into a set of discrete sub-domains, usually called elements.

FEA is a computer-based process used for modeling complex products and systems. These simulations occur in a virtual environment for the purpose of 'solving' or finding a series of solutions to potentially complex performance issues. It is useful for all degrees of science[3]

Specifically, as applied to material sciences and engineering, it is used to calculate the strength and behavioral characteristics of a material under conditions such as stress, vibration and deflection. It can be used to investigate large-scale and small-scale behaviors of materials. FEA software is now faster and easier to use than ever before. It is no longer available from only a few companies and comes in a variety of levels of complexity. It is now so diverse that it can be spread around manufacturing facilities and even embedded into other design and control software that is already being used by operators

BASIC CONCEPT

The finite element method (FEM) is a numerical method for solving problems of engineering and mathematical physics. It is also referred to as finite element analysis (FEA). Typical problem areas of interest include structural analysis, heat transfer, fluid flow, mass transport, and electromagnetic potential. The analytical solution of these problems generally require the solution to boundary value problems for partial differentialequations. The finite element method formulation of the problem results in a system of algebraic equations. The method yields approximate values of the unknowns at discrete number of points over the domain

A major advantage of finite element analysis (FEA) is its ability to solve complex biomechanical problems for witch other study methods are inadequate. Stress, strain and some other qualities can be calculated in every point throughout the structure. FEA is also being used as part of the design process to simulate possible structure failure, as a mean to reduce the need for making prototypes, and reducing a need for performing actual experiments, that are usually expensive and time-consuming. This method allows researchers to overcome some ethical and methodological limitations and enables them to verify how the stresses are transferred throughout the materials.

The procedure consists of three steps

i) Pre-processing phase
ii) Solution phase
iii) Post-processing phase'

In the first phase the structure or region to be studied is made to undergo a CT scan.

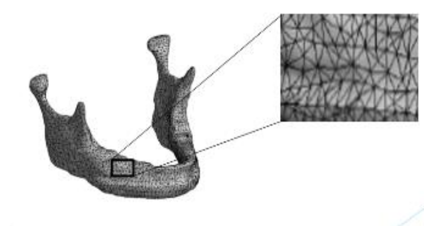

Figure 1 shows a finite element model of a mandible along with a magnified segment showing triangular elements connected to each other at nodes[4].

Using the appropriate software a finite element model of the region is created with the help of the CT scan.

The model is made up of ***elements*** which are triangular in shape, each of which are connected to each other at junction known as ***nodes.***

In the second phase, we Boundary condition is defined in this phase.

Boundary conditions means that suppose an element is constructed on the computer and a force is applied to it, it will act like a free-floating rigid body and will undergo a translatory or rotatory motion or a combination of the two without experiencing deformation.

To study its deformation, some degrees of freedom must be restricted (movement of the node in each direction x, y, and z) for some of the nodes. Such constraints are termed boundary conditions.

The third phase includes result output obtained via the processing phase.Output can be achieved via following three different manners

a) Graphical output
b) Numeric output
c) Animated output

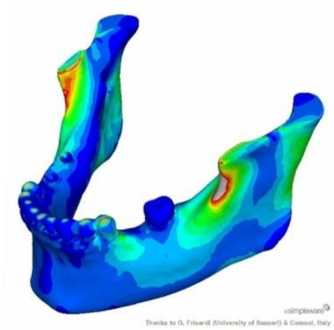

Figure 2 illustrates a graphical output with red colour showing the area of maximum stress and blue colour showing the area of minimum stress[5].

IMPORTANCE OF FINITE ELEMENT STUDY IN DENTISTRY

Biomechanics is fundamental to any dental practice, including dental restorations, movement of misaligned teeth, implant design, dental trauma, surgical removal of impacted teeth, and craniofacial growth modification. Following functional load, stresses and strains are created inside the biological structures. Stress at any point in the construction is critical and governs failure of the prostheses, remodeling of bone, and type of tooth movement. However, *in vivo* methods that directly measure internal stresses without altering the tissues do not currently exist.

APPLICATION OF FINITE ELEMENT METHOD IN DENTISTRY

Deformation and stresses are generated when loads are applied to a structure. This is usual, and is how a structure performs its structural function. But if stresses become excessive and exceed the elastic limit, structural failure may result[6].

Prosthodontics

FEM has been shown to be a useful tool when investigating complex systems that are difficult to standardize during in vitro and in vivo studies. It has been used to evaluate the influence of the type of material (metal, carbon, glass fiber and zirconia ceramic) and the external configuration of the dowel (smooth and serrated) on the stress distribution of teeth restored with varying dowel systems.

The static loading such as bite forces is usually applied as point forces to study prosthetic designs and dental restorations. The bite force, however, presents huge variations (both magnitude and direction) based on previous experimental measures (Proffit et al., 1983; Proffit and Field 1983).[7,8] Fortunately, FEA allows for easy changes in force magnitudes and directions to approximate experimental data, which can serve as a reasonable parametric study to assess different loading effects.

Restorative Dentistry

FEM has been used to analyze stresses generated in teeth and restorations. It has proven to be a useful tool for understanding tooth biomechanics and the biomimetic approach in restorative dentistry.

Orthodontics

FEM and Arch Wire Activation:

Canales et al[9] birth death technique in orthodontic brackets proved to be a biomechanicalstimulation for placement of continuous arch wires. De Oliveira BF et al[10]established that greater tooth displacement was found in shortened dental arches than in complete dentalarches. An increase in arch perimeter was associated with mandibular lateral expansion.[11]Inter molar expansion of 1mm increased the arch perimeter by 0.30mm

Tominagaet al[12] used a three dimensional finite element analysis in sliding mechanics to simulate en masse anterior tooth retraction. On the application of retraction force the displacement of maxillary incisor and arch wire deformation were calculated. An x et al[13] investigated the effect of en masse retraction of maxillary teeth and established that excessive retraction can be avoided and intrusion and torque control can be achieved

FEM and Alveolar Bone:
Wang H et al[14] established 3-D finite element model of maxillary first molars and the stress magnitude and distribution within the Periodontal ligament of maxillary first molars was calculated when loaded with intrusion force.

FEM and T-Loops:
The commonly analyzed parameter when segmental T-loop was used for canine retraction is ideal moment to force ratio. The load system was significantly affected due to clinical changes in canine position and angulation. The highest moment to force ratio (8.5-9.3) was showed by upright Opus loops and l-loops, when the loops were centred on the caninebrackets.

FEM and Trans Palatal Bar and Twin Block:
Transpalatal arch design was used to optimize unilateral molar rotation correction using a finite element method by Geramy A et al.[15] An activation of about 0.1 and 1.0 mm produced the same increasing patterns regarding the energy levels.

FEM and Root Resorption:
The initial effects of stress on the periodontal ligament was compared over time in orthodontic external root resorption, necrosis and TRAP+cell population.[16]

FEM and Expanders:
Araugio RM et al [17]analysed that less dental tipping was generated when the ideal screw position is slightly above the maxillary first molar center of resistance.

FEM in Orthognathic Surgery:
The hybrid technique fixation of the Sagittal split ramus osteotomy was evaluated by SatoFR et al[18] for their mechanical characteristics and stress distribution.

FEM in Individual Tooth Movements:
Van Schepdaelet al[19] analyzed and concluded that the differences between analytical and FEM results were small except at the alveolar crest region, but a success in the global behaviour of PDL was observed. Vikram NR et al[20] established an increase in cementumthicknesss when the apical stress induced in the periodontal ligament decreased.A stress in the cementum and Periodontal ligament occurred as a result of clinical delivery of anorthodontic force. Lin YL et al found that there was a higher stress level and homogenous stress distribution in individual teeth with posterior cross bite than teeth with normal occlusion. Geramy A analyzed a method to move palatally erupted lateral incisors labially. Equal forces of about 0.15N were applied. An intrusive component could also be added.

FEM in Brackets:
The bracket design had less influence on the torquing moment. An increased torque control capability was exhibited by wider brackets.[21]Holberg C et al attempted to create an anisotropic finite element method model of mandibular bone and orthodontic bracket[22]The results indicated that the risk of enamel fracture depended on the individual debondingprocedure.

FEM in Tip Edge Force System:
Zhang Y et al analyzed the distal and extrusive displacement of maxillary first molar under the effect of tip back bend, which could be controlled by the precise control of the degree and position of tip back bend.[23]

FEM in Soft Tissues:
Chen S et al did a study to get individualized facial three dimensional FE model for prediction of treatment related morphological change of facial soft tissue. He found an average deviation of 0.47mm and 0.75mm in the soft and hard tissue respectively.[24]

FUTURE IMPLICATION OF FINITE ELEMENT METHOD IN DENTISTRY WITH SPECIAL CONCERN TO ORTHODONTICS

Although FEA techniques have greatly improved over the past few decades, further developments remain. More robust solid models with increased capability to manipulate CAD objects would allow increased research in this area.

The ability to fix minor problematic geometry and easily create models with minor variations would greatly reduce the time required to model different biomechanical situations. Additionally, adding frictional boundaries conditions between teeth and active ligations for orthodontic appliances will continue to increase the accuracy of these models.

Three dimensional dynamic simulations for assessing tooth injury, similar to those demonstrated in 2D studies should be reevaluated. While techniques will continually be optimized to improve numerical approximations, this does not negate the value of finite element techniques in dentistry.

These techniques use proven engineering principles to model aspects of dentistry that are unable to be efficiently investigated using clinical techniques, and will continue to provide valuable clinical insights regarding dental biomechanics

References

1) Zienkiewicz, O., Kelly, D. and Bettess, P. (1977). The coupling of the finite element method and boundary solution procedures. *International Journal for Numerical Methods in Engineering*, 11(2), pp.355-375.

2) Hughes, T. (2000). The finite element method. Mineola, NY: Dover Publications.

3) En.wikipedia.org. (2018). Finite element method. [online] Available at:https://en.wikipedia.org/wiki/Finite_element_method#The_structure_of_finite_element_methods [Accessed 17 Jul. 2018].

4) Hung Tsai M. Research on Biomechanical Behavior of Temporomandibular Joint Disorders Using Occlusal Splint Treatment: Computed-Aided Analysis. Journal of Biomedical Sciencies. 2015;04(02).

5) Dental: Finite element analysis of a 3D lower jaw model. | FEM | Finite element method, Mechanical engineering, Physics [Internet] Pinterest. 2019 [cited 14 October 2019] Availablefrom:https://www.pinterest.com/pin/326722147941933810/

6) Liu, C., Zhu, X. and Zhang, X. (2015). Three-dimensional finite element analysis of maxillary protraction with labiolingual arches and implants. *American Journal of Orthodontics and Dentofacial Orthopedics*, 148(3), pp.466-478.

7) Proffit WR, Fields HW, Nixon WL. Occlusal forces in normal and long face adults.J Dent Res 1983; 62:566-571.

8) Proffit WR, Fields HW. Occlusal forces in normal and long face children. J Dent Res 1983; 62:571-574.

9) Canales C, Larson M, Grauer D, Sheats R, Stevens C, Ko CC. A novel biomechanical model assessing continuous orthodontic archwireactivation.Am J OrthodDentofacialOrthop 2013; 143:281-190.

10) De Oliveira BF, Seraidarian PI, de Oliveira SG, Landre J Jr, Pithon MM, Oliveira DD. Tooth displacement in shortened dental arches: a three-dimensional finite element study. J Prosthet Dent. 2014; **111**(6):460-465

11) Baswaraj, Hemanth M, Jayasudha, Patil C, Sunilkumar P, Raghuveer HP, Chandralekha B. An experimental study of arch perimeter and arch width increase with mandibular expansion: a finite element method. J Contemp Dent Pract.2013; 14:104-110.

12) Tominaga JY, Chiang PC, Ozaki H, Tanaka M, Koga Y, Bourauel C, Yoshida N. Effect of play between bracket and archwire on anterior tooth movement in sliding mechanics: A three-dimensional finiteelementstudy.J Dent Biomech 2012; **3**:587-612.

13) An X, Chen H, Si Q, Zhou Y, Liu B, Wang J. A three dimensional finite element analysis on en-masse retraction of maxillary anterior teeth by rocking-chair archwireinsliding mechanics. Hua Xi Kou Qiang Yi XueZaZhi. 2013; 31:21-25.

14) Wang H, Wu JY, Zhou Q, Liu J.Initial stress in the periodontal membrane of maxillary first molar with different alveolar bone height by intrusion: 3-dimensional finite element analysis. Shanghai Kou Qiang Yi Xue 2013; 22:247-251.

15) Geramy A, Etezadi T. Optimization of unilateral molar rotation correction by atranspalatal bar: a threedimensional analysis using the finite element method. J Orthod 2013; 40:197-205.

16) Viecilli RF, Kar-Kuri MH, Varriale J, Budiman A, Janal M. Effects of initial stresses and time on orthodontic external root resorption. J Dent Res 2013; 92:346-35.

17) Araugio RM, Landre J Jr, Silva Dde L, Pacheco W, Pithon MM, Oliveira DD. Influence of the expansion screw height on the dental effects of the hyrax expander: a study with finite elements. Am J OrthodDentofacialOrthop 2013; 143:221-7.

18) Sato FR, Asprino L, Consani S, Noritomi PY, de Moraes M. A comparative evaluation of the hybrid technique for fixation of the sagittal split ramus osteotomy in mandibular advancement by mechanical, photoelastic, and finite element analysis.OralSurg Oral Med Oral Pathol Oral Radiol. 2012; 114:60-68

19) Van Schepdael A, Geris L, Vander Sloten J. Analytical determination of stress patterns in the periodontal ligament during orthodontic tooth movement. Med EngPhys 2013; 35:403-410.

20) Vikram NR, Senthil Kumar KS, Nagachandran KS, Hashir YM. Apical stress distribution on maxillary central incisor during various orthodontic tooth movements by varying cemental and two different periodontal ligament thicknesses: a FEM study. Indian J Dent Res 2012; 23:213-220.

21) Huang Y, Keilig L, Rahimi A, Reimann S, Bourauel C. Torque capabilities of selfligating and conventional brackets under the effect of bracket width and free wire length. OrthodCraniofac Res 2012; 15:255-262.

22) Holberg C, Winterhalder P, Holberg N, Wichelhaus A, Rudzki-Janson I. Orthodontic bracket debonding: risk of enamel fracture.Clin Oral Investig 2014; 18:327-334.

23) Zhang Y, Xia X, Deng F, Fan Y, Wang C, Huang Q. Analysis of tip back bend on initial displacement of the maxillary first molar in Tip-Edge force system by three-dimensional finite element method.Hua Xi Kou Qiang Yi XueZaZhi 2012; 30:329-332.

24) Chen S, Xu TM, Lou HD, Rong QG. Individualized three dimensional finite element model of facial soft tissue and preliminary application in orthodontics. Zhonghua Kou Qiang Yi XueZaZhi 2012; 47:730-734.