IOSR Journal of Dental and Medical Sciences (IOSR-JDMS)
e-ISSN: 2279-0853, p-ISSN: 2279-0861.Volume 15, Issue 3 Ver. III (Mar. 2016), PP 38-41
www.iosrjournals.org

Finite Element Method: An Overview

Dr.Falguni Mehta[1], Dr.Hrishabh Joshi[2]
1(Head of Department, Department of Orthodontics, Govt. Dental College and Hospital, Ahmedabad)
2(3rd Year Post Graduate, Department of Orthodontics, Govt. Dental College and Hospital, Ahmedabad)

Abstract: *The finite element method (FEM) involves a series of mathematical computational procedures to calculate the load distribution in each element.Such a structural analysis allows the determination of stress resulting from external force, pressure, thermal change, and other factors. This method is extremely useful for indicating mechanical aspects of biomaterials and human tissues that can hardly be measured in vivo. The results obtained can then be studied using visualization software within the FEM environment to view a variety of parameters, and to fully identify implications of the analysis. This paper shows the basic concept, methodology, advantages, limitations and applications of finite element method (FEM) in dentistry.*
Keywords**:** *Finite element method, finite element analysis, Nodes, Geometric model, meshing*

I. Introduction

The majority of previous studies done for elucidation of biomechanical behavior are limited in their evaluation of stress distributions in the internal structures.Thus because of difficulties in measuring strains and stresses in living tissues complete understanding of complex mechanisms have not been understood properly.
Finite element analysis makes it possible to elucidate biomechanical components such as stresses and strains induced in the living structures from various external forces[1]. The finite element method of stress analysis (FEM) is a computer-aided mathematical technique for obtaining approximate numerical solutions to the abstract equations of calculus that predict the response of physical systems subjected to external influences[2,3]. FEM allows stress distributions and levels to be evaluated in systems with irregular geometry and nonhomogeneous physical properties.The finite element method has proven to be a useful tool in a number ofbiologicalapplications. The mathematical limitations inherent in linear methods from plane geometry were described accurately many years ago by Moyers and Bookstein. This method generates a mathematical transformation related to both size and shape. The use of transformations in this manner accomplishes a greater degree of sophistication for the analysis of craniofacial growth and development.
The finite element method (FEM) is a highly precise technique used to analyze structural stress. Used in engineering for years, this method uses the computer to solve large numbers of equations to calculate stress on the basis of the physical properties of structures being analyzed [4,5]. FEM has many advantages over other methods (such as the photoelastic method), highlighted by the ability to includeheterogeneity of tooth material and irregularity of the tooth contour in the model design and the relative ease with which loads can be applied at different directions and magnitudes for a more complete analysis. Finite element analysis has been used in dentistry to investigate a wide range of topics, such as the structure of teeth, biomaterials and restorations, dental implants, and root canals [6-8].

II. Literature review

Finite element method was originally developed in 1956 in the aircraft industry. This technique has since been in widespread use not only in aerospace engineering, but also in civil engineering. The finite element method has proved to be extremely effective for the treatment of problems of plane stress and plane strain.
Yettram et al were amongst the first to employ a two-dimensional finite element model of a maxillary central incisor to determine the instantaneous centre of rotation of this tooth during translation[9]. First three dimensional FEM study in dentistry appeared in 1974, where J.W..Farah and R.G.Craigdid finite element stress analysis in a restored axisymmetric first molar[10]. Moss et al described the application of Finite element methods for quantitative descriptions of cranial skeletal size and shape change with local growth significance, independent of any external frame of reference, and, by so doing, eliminating the principal source of methodological error in customary roentgenographiccephalometry[4].Using more complex three dimensional models Wilson et al,Tanne et al. and McGuinness et al havestudied moment to force ratios and stress distributions during orthodontic tooth movement. Cobo et al. studied periodontal Stresses during tooth movement[12-14].In the field of Dentofacial Orthopedics, Finite Element models have been employed to evaluate the stress distribution induced within the craniofacial complex during the application of protraction headgear, orthopedic chin cup forces and conventional headgear forces[15-19]. Kazuo Tanne et al investigated stress distributions in the craniofacial complex by means of the finite element analysis[20].Ghosh et alhave used

three-dimensional FEM models of ceramic orthodontic bracket designs to determine the stress distribution and likely mode of cohesive failure within the bracket when a full dimension stainless steel arch wire is engaged within the bracket slot [21].Similarly, Rossouw and Tereblanche have used a simplified three dimensional finite element model to evaluate the stress distribution around orthodontic attachments during debonding[22] .

Ghosh and Nanda generated finite element models for selected ceramic brackets and graphically display the stress distribution in the brackets when subjected to arch wire torsion and tipping forces[21].Katona compared different methods of bracket removal and suggested that different loading methods resulted in significantly different stress patterns[23].Halazonetisused two dimensional model to determine periodontal ligament (PDL) stress distribution following force application at varying distances from the centre of resistance of a maxillary incisor[24].Bobak et al used finite element method of analysis (FEM) to analyze theoretically the effects of a transpalatal arch (TPA) on periodontal stresses of molars that were subjected to typical retraction forces[25].

G.D. Singh, J.A. McNamara traced cephalographs and color-coded finite element (FEM) analysis was used to measure differences in morphology[26].Iseri et al evaluated the biomechanical effect of rapid maxillary expansion (RME) on the craniofacial complex by using a three-dimensional finite element model (FEM) of the craniofacial skeleton.Jafari et al analyzed the stress distribution patterns within the craniofacial complex during rapid maxillary expansion. A finite element model of a young human skull was generated using data from computerized tomographic scans of a dried skull[27].Lin et aldid a study on microimplants and concluded that increased exposure lengths resulted in higher bone stresses adjacent to the mini-implant. The direction of orthodontic force had no significant effect on cortical bone stress[28].Anshul Chaudhary et al(2015) did a fixed functional appliance study by finite element model analysis which caused increase in the maximum principal stress and the von Mises stress in both the cortical bone and the condylar region of the mandible by more than 2 times[29].

III. Basic steps involved in carrying out FEA are [30]:

- Pre-processing.
- Conversion of geometric model into finite element model.
- Assembly/Material Property data representation.
- Defining the boundary conditions.
- Loading Configuration.
- Processing.
- Post-processing.

III.1.Preprocessing: Construction of the Geometric model:

The purpose of the geometric modeling phase is to represent geometry in terms of points, lines, areas and volume. Complicated or smooth objects can be represented by geometrically simple pieces.

III.2.Conversion of Geometric model to Finite Element Model:

Discretization is the process of dividing problem into several small elements, connected with nodes. All elements and nodes must be numbered so that a setup of matrix connectivity is established. This greatly affects the computing time. The elements could be one, two or three-dimensional and in various shapes. It is essential that the elements are not overlapping but 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'.

III.3.Assembly / Material Property data representation:

Equations are developed for each element in the FEM mesh and assembled into a set of global equations that model the properties of the entire system. Minimum material properties required are poisons ratio and young's modulus.

III.4.Defining the Boundary Conditions:

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.

III.5.Loading configuration:

Application of force at various points of geometry and its configuration.

III.6.Processing:

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

III.7.Post-Processing:

The output 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 solid mechanical problems, the output is nodal displacement and element stresses. *Graphic outputs* and displays are usually more informative. The curves and contours of the field variable can be plotted and displayed. Also deformed 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.

IV. Applications in orthodontics

1. Finite element analysis has been applied to the description of form changes in biological structures (morphometrics), particularly in the area of growth and development[31].
2. Finite element analysis as well as other related morphometric techniques such a macro-element and the boundary integral equation method (BIE) is useful for the assessment of complex shape changes[32].
3. Analysis of stresses produced in the periodontal ligament when subjected to orthodontic forces[33-34].
4. Finite element method is also useful for structures with inherent material homogeneity and potentially complicated shapes such as dental implants[35-37].
5. The type of predictive computer model described may be used to study the biomechanics of orthodontic tooth movement, whilst accurately assessing the effect of new appliance systems and materials without the need to go to animal or other less representative models.
6. The mechanical behavior of the orthodontic wires and different design of brackets and its contact problem can be well modeled and simulated by the finite element method. This method is an important tool in the development and improvement of orthodontic bracket and wires design[38].

V. Advantages of FEM

1. Extensive instrumentation is not required.
2. Complex larger problems can be split into smaller problems.
3. It is a completely non-invasive procedure.
4. Three dimensional models can be generated.
5. Actual physical properties can be simulated and external environment can be simulated.
6. The operator can repeat the study as many times as possible.

VI. Disadvantages of FEM

1. The tooth is treated as pinned to the bone via periodontal ligament. This rigidity in the elemental nodal complex can result in errors in calculations.
2. To simulate physical environment certain assumptions are made which can result in errors in maximum stress calculations.
3. The properties assigned for calculations are not satisfactory so better defined properties must be used to get more precise results.

References

[1]. Burnett DS. Finite Element Analysis—From Concepts to Applications. Boston, Mass.: Addison-Wesley; 1987.

[2]. Moyers RE, Bookstein FL. The inappropriateness of conventional cephalometrics. *Am J Orthod 1979; 75*: 599-617.

[3]. Bookstein FL. Measuring treatment effects on craniofacial growth. In Carison DS, ed: Clinical alteration of the growin face. Ann Arbor, MI: Center for Human Growth and Developement, University of Michigan, 1983: 65-80.

[4]. Rubin C, Krishnamurthy N, Capilouto E, Yi H. Stress analysis of the human tooth using a three-dimensional finite element model. *J Dent Res. 1983; 62*: 82–86.

[5]. Darendeliler S, Darendeliler H, Kinoglu T. Analysis of a central maxillary incisor by using a three-dimensional finite element method. *J Oral Rehabil. 1992; 19*: 371–383.

[6]. Akpinar I, Anil N, Parnas L. A natural tooth's stress distribution in occlusion with a dental implant. *J Oral Rehabil. 2000; 27*: 538–545.

[7]. Holmgren EP, Seckinger RJ, Kilgren LM, Mante F. Evaluating parameters of osseointegrated dental implants using finite element analysis—a 2-dimensional comparative study examining the effects of implant diameter, implant shape, and load direction. *JOral Implantol. 1998; 24*: 80–88.

[8]. Papavasiliou G, Kamposiora P, Bayne SC, Felton DA. Three-dimensional finite element analysis of stress-distribution around single tooth implants as a function of bony support, prosthesis type, and loading during function. *J Prosthet Dent. 1996; 76*: 633–640

[9]. YettramAL,WrightKWJ,HoustonWJB.Center of rotation of a maxillary central incisor under orthodontic loading.*Br J Orthod 1977*

[10]. Farah, J.W., Craig, R.G., Sikarskei, D.L. Photoelastic and finite element stress analysis of a restored axisymmetric first molar. *J Biomech. 1973;6*:511–520.

[11]. Melvin L. Moss, Richard Skalak, Himanshu Patel, Kasturi Sen, Letty MossSalentijn, Masanobu Shinozuka, Henning Vilmann. Finite element method modeling of craniofacial growth.*Am J OrthodDentofacialOrthop. 1985.*

[12]. Wilson AN, Jones ML, Middleton J. The effect of the periodontal ligament on bone remodelling. In: Middleton J, Jones ML, Pande GN, eds. Recent Advances in Computer Methods in Biomechanics and Biomechanical Engineering, Books and Journals International. Swansen, UK: *Books and Journals International; 1992*;150– 158.

[13]. McGuiness NJP, Wilson A, Jones ML, Middleton J, et al. Stress induced by edgewise appliances in the periodontal ligament—a finite element study. *Angle Orthod. 1991*

[14]. Kazuo Tanne, Mamoru Sakuda, Charles J. Burstone. Three-dimensional finite element analysis for stress in the periodontal tissue by orthodontic forces, *Am J OrthodDentofacialOrthop. 1987*

[15]. Tanne K, Miyasaka J, Yamagata Y, et al. Three dimensional model of the human craniofacial skeleton: method and preliminary results using finite element analysis. *J Biomechl Eng. 1988;10:* 246–252.

[16]. Tanne K, Hiraga J, Sakuda M. Effects of directions of maxillary protraction forces on biomechanical changes in craniofacial complex.*Eur J Orthod. 1989;11*:382–391

[17]. Tanne K, Hiraga J, Kakiuchi K, et al. Biomechanical effect of anteriorly directed extra-oral forces on the craniofacial complex: A study using the finite element method. *Am J OrthodDentofacOrthop. 1989;95*:200–207

[18]. Tanne K, Lu YC, Tanaka E, Sakuda M. Biomechanical changes of the mandible from orthopaedic chin cup force studied in a three dimensional finite element model. *Eur J Orthod. 1993;15*:527– 533.

[19]. Miyasaka-Higari J, Tanne K, Nakamura S. Finite element analysis for stresses in the craniofacial sutures produced by maxillary protraction forces applied at the upper canines. *Br J Orthod. 1994; 21*:343–349.

[20]. Tanne K, Matsubara S. Association between the direction of orththopaedic headgear force and sutural responses in the nasomaxillary complex. *Angle Orthod. 1996;66*:125–130.

[21]. Joydeep Ghosh, Ram S. Nanda, Manville G. Duncanson Jr., G.Fräns Currier Ceramic bracket design: An analysis using the finite element method. *Am J OrthodDentofacial Orthop1995;108(6)*:575-582

[22]. Rossouw PE, Tereblanche E. Use of the finite element analysis in assessing stress distribution during de-bonding. *J ClinOrthod. 1995;29*:713–717.

[23]. Katona TR. Stresses developed during clinical de-bonding of stainless steel orthodontic brackets. *Angle Orthod. 1997;67*:39– 46

[24]. Halazonetis DJ. Computer experiments using a two dimensional model of tooth support. *Am J OrthodDentofacOrthop. 1996; 109:*598–606.

[25]. VoytekBobak, Richard L Christiansen, Scott J Hollister, David H Kohn ; stress related molar responses to trans palatal arch : a finite element analysis. *AJODO; Sep 1997.*

[26]. G. D. Singh, J. A. McNamara *Jr*., and S. Lozanoff (*1998*) Mandibular morphology in subjects with Class III malocclusions: Finite-element morphometry. *Angle Orthod* 1998*; 68(5)*: 409-418.

[27]. Jafari, A., Shetty, K.S., Kumar, M. Study of stress distribution and displacement of various craniofacial structures following application of transverse orthopedic forces—a three-dimensional FEM study. *Angle Orthod. 2003; 73*: 12–20.

[28]. Lin TS, Tsai FD, Chen CY, Lin LW. Factorial analysis of variables affecting bone stress adjacent to the orthodontic anchorage mini-implant with finite element analysis. *Am J OrthodDentofacialOrthop. 2013; 143*:182-189.

[29]. Chaudhry, A., Sidhu, M.S., Chaudhary, G., Grover, S., Prabhakar, M., Malik, V. Evaluation of stress changes in mandible with twin block appliance—a finite element study. *Baba Farid University Dental Journal. 2014; 5:*13–20.

[30]. Prasad Konda, Tarannum SA, Basic principles of finite element method and its applications in orthodontics*JPBMS, 2012, 16* (11).

[31]. Melvin L. Moss, Richard Skalak, Himanshu Patel, Kasturi Sen, Letty Moss- Salentijn, Masanobu Shinozuka, Henning Vilmann. Finite element method modelling of craniofacial growth.*Am J OrthodDentofacialOrthop. 1985; 87(6):* 453-472.

[32]. Stephanie R. Toms, Alan W. Eberhardt. A nonlinear finite element analysis of the periodontal ligament under orthodontic tooth loading. *Am J OrthodDentofacialOrthop 2003:123(6)*:657-665

[33]. Gallas MM, Abeleira MT, Fernández JR, Burguera M. Three-dimensional numerical simulation of dental implants as orthodontic anchorage.*Eur J Orthod. 2005 Feb; 27(1)*: 12-16.

[34]. Lin Jiang, Liang Kong, Tao Li , ZexuGu, RuiHou , YinzhongDuan. Optimal selections of orthodontic mini-implant diameter and length by biomechanical consideration: A three-dimensional finite element analysis. *Advances in Engineering Software. 2009; 40 (11)*; 1124-1130.

[35]. K.R. Williams, J.T. Edmundson. Orthodontic tooth movement analysed by the Finite Element Method. *Biomaterials 1984; 5(6)*: 347-351.

[36]. Jones ML, Hickman J, Middleton J, Knox J, Volp C. A validated finite element method study of orthodontic tooth movement in the human subject. *J Orthod 2001; 28:* 29–38.

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

[38]. Joydeep Ghosh, Ram S. Nanda, Manville G. Duncanson Jr., G.Fräns Currier Ceramic bracket design: An analysis using the finite element method. *Am J OrthodDentofacial Orthop1995; 108(6)*: 575-582.