EUROPEAN JOURNAL OF BIOMEDICAL AND PHARMACEUTICAL SCIENCES

http://www.ejbps.com

ISSN 2349-8870
Volume: 6
Issue: 11
167-170
Year: 2019

FINITE ELEMENT ANALYSIS (FEA) WITH ITS METHODOLOGY AND USES IN ORTHODONTICS AND DENTOFACIAL ORTHOPAEDICS: AN OVERVIEW

Mehak Bansal[1], Dhiraj Garg[2]* and Vinni Arora[3]

[1]Post Graduate, Department of Orthodontics and Dentofacial Orthopaedics, Post Graduate Institute of Dental Sciences, Pandit Bhagwat Dayal Sharma, University of Health Sciences, Rohtak, Haryana, India.
[2]Post Graduate, Department of Orthodontics and Dentofacial Orthopaedics, Maulana Azad Institute of Dental Sciences, New Delhi, India.
[3]Senior Resident, Department of Orthodontics and Dentofacial Orthopaedics, Post Graduate Institute of Dental Sciences, Pandit Bhagwat Dayal Sharma, University of Health Sciences, Rohtak, Haryana, India.

***Corresponding Author: Dr. Dhiraj Garg**
Post Graduate, Department of Orthodontics and Dentofacial Orthopaedics, Maulana Azad Institute of Dental Sciences, New Delhi, India.

Article Received on 14/08/2019 **Article Revised on 03/09/2019** **Article Accepted on 24/09/2019**

ABSTRACT

Introduction: In attempt to improve and evaluate the mechanical effects of various orthodontics appliances and techniques, analysis of stress and strain of the same under various loading circumstances has become an integral part of research in recent era. As we all know, our skull consists of various complex structures with a very limited accessibility. Due to this, most biomechanical research of the maxillofacial and oro-dental environment has been performed in vitro. Finite Element Analysis (FEA) is a modern engineering resource for numerical stress analysis, with an advantage of being applicable to solids of irregular geometry that contain heterogeneous material properties. **Objective:** This article aims at reviewing and discussing the methodology of the finite element method application and its applicability in Orthodontics and dentofacial orthopaedics. **Results:** FEM is able to calculate the stress distribution at the interface between periodontal ligament and alveolar bone, and the shifting trend in various types of tooth movement when using different types of orthodontic devices. Therefore, it is needed to know specific software for this purpose. **Conclusions:** FEM is an important experimental method to answer questions about complex biomechanics of orthodontic and orthopaedic appliances, overcoming the disadvantages of other experimental approaches.

KEYWORDS: Bioengineering, Finite element method, Orthodontics, Literature review.

INTRODUCTION

Finite element method is a computer-aided numerical technique for finding approximate solutions to physical systems subjected to external influences under specific boundary conditions in order to solve complex problems in engineering, science and applied mathematics.[1] R. Courant in 1943 originally developed this technique to investigate torsion on a cylinder, its uses has now been expanded to analyze thermal, electromagnetic, fluid, and structural working environments.[2] The basic foundation of FEM is that a large complex geometry can be subdivided into simpler parts in order to accurately represent the complex geometry while including the dissimilar material properties and capturing the local effects.[3] The finite element method (FEM) associated with modern computer technology is a stress analysis method.[2] The basic principle is to solve the stress distribution value of the research object by applying the elastic theory. Compared with the traditional experimental stress analysis method, the advantages and characteristics of FEM are that it can simulate the stress and displacement trend of any part through the established model; by converting the biomedical model into a mathematical mechanical model, the same model can be used repeatedly for various loadings. The calculation of the situation ensures that the research model is completely similar; the processing of huge data by computer, the calculation efficiency is high, the result precision is high, and the human and material resources are greatly saved. At present, FEM is more and more in the research of orthopaedics biomechanics. Important position, it is widely believed that this is one of the most advanced and effective methods of biomechanical research, and its superiority is dependent on the establishment of a dynamic three-dimensional finite element model with biomechanical properties.

METHOD

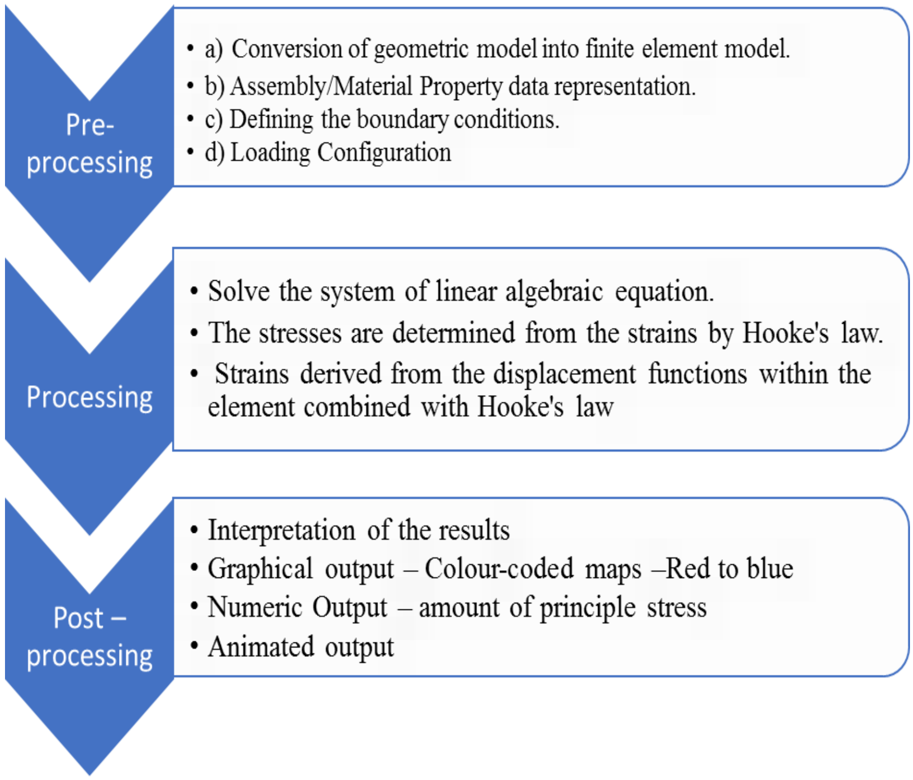
A. Computed Tomography (CT) Raw Data

CBCT scan of a skull or a volunteer will be obtained for obtaining the DICOM raw data. The CBCT image will be saved as DICOM (digital imaging and communication in medicine) data and exported to 3D image processing and editing software. The 3D finite model will be created using appropriate FEA Software.

B. Finite Element Analysis

Basic steps involved in carrying out FEA are

1) Pre-processing.
   a) Conversion of geometric model into finite element model.
   b) Assembly/Material Property data representation.
   c) Defining the boundary conditions.
   d) Loading Configuration.
2) Processing.
3) Post-processing.



1. Pre-processing

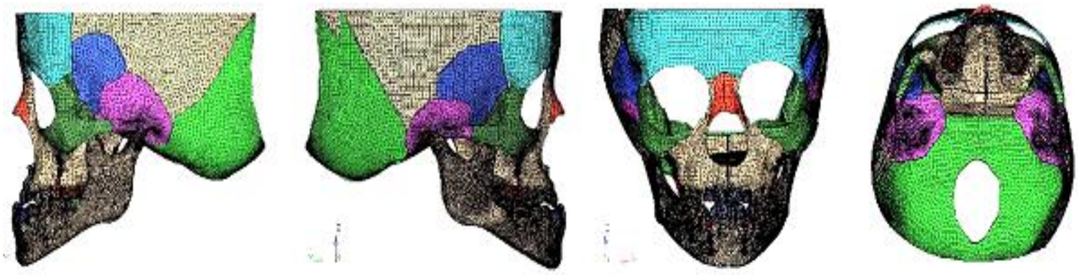
Construction of the Geometric model

The purpose of the geometric modelling phase is to represent geometry in terms of points, lines, areas and volume. Complicated or smooth objects can be represented by geometrically simple pieces (Elements). This can be achieved by: - 3D – CT scanner: Usually done for modelling complex structures or living tissues. For example, craniofacial skeleton, maxilla or mandible.

A. 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'. Figure 1 illustrates a lateral, frontal and basal view of a 3-D meshed finite element model.

Lateral, frontal and basal view of a 3-D meshed FEM



B. 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. Table 1 showing ideal physical properties of the dentofacial structures and materials.[4,5]

Table I: Young's modulus and Poisson's ratio for the Dentofacial structures and materials.

Material	Young's modulus (MPa)	Poisson's ratio
Cortical bone	1.37×10^4	0.30
Cancellous bone	7.9×10^3	0.30
Miniplate	1.05×10^5	0.33
Miniscrew	1.05×10^5	0.33
Suture	7	0.40
Tooth	2.07×10^4	0.30
PDL	50.00	0.49

C. 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.

D. Loading configuration

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

2. 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.

Calculation of Principle Stresses

$$\boldsymbol{\sigma} = \begin{bmatrix} \sigma_{11} & \sigma_{12} & \sigma_{13} \\ \sigma_{21} & \sigma_{22} & \sigma_{23} \\ \sigma_{31} & \sigma_{32} & \sigma_{33} \end{bmatrix} \equiv \begin{bmatrix} \sigma_{xx} & \sigma_{xy} & \sigma_{xz} \\ \sigma_{yx} & \sigma_{yy} & \sigma_{yz} \\ \sigma_{zx} & \sigma_{zy} & \sigma_{zz} \end{bmatrix} \equiv \begin{bmatrix} \sigma_x & \tau_{xy} & \tau_{xz} \\ \tau_{yx} & \sigma_y & \tau_{yz} \\ \tau_{zx} & \tau_{zy} & \sigma_z \end{bmatrix}$$

The cubic equation above is used to calculate all principle stresses within each node in the model. The determinant is expanded, producing an equation, before derivation of three roots. The most positive root is defined as σ1, which represents first principle stress (tensile), while the most negative value is defined as σ3 representing third principle stresses (compressive). The second principle stress usually approaches zero, and is not significant for the study.

3. Post-Processing

The output from the Finite Element Analysis is primarily in the numerical form. It usually consists of nodal values 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.

ADVANTAGES AND LIMITATIONS

Advantages

1. FEM is a non-invasive technique.
2. As it is less time consuming, so the complicated studies can now be evaluated in a lesser time frame.
3. It can be applicable to solid and fluid structural interactions as well as linear and non-linear.
4. Complex problems can be split into smaller number of problems.
5. Dynamic and static analysis can be done.
6. The physical properties involved are not affected during reproducibility.
7. Stereo lithographic models can be replaced with FEA for presurgical planning, providing an economical solution for the same.
8. It's very easy to simulate any biological condition in pre, intra and post-treatment stages by using FEA, to achieve more accurate and reliable results.
9. Wide range instrumentation requirement are less.
10. Repetitions of study can be possible as many times as the operator wants.

Limitations

1. Imprecise data, information, and interpretation would lead to entirely misguiding results.
2. Complex anatomy and lack of entire knowledge about human structures mechanical behaviours, modelling becomes extremely difficult.
3. Results will depend on the personnel involved in the process as certain assumptions are bound to be accepted.
4. The progress in the FEA will be limited until better defined physical properties for enamel, dentin and periodontal ligament and cancellous and cortical bone are available.
5. Some assumption will introduce some error i.e. the tooth is treated as pinned to the supporting bone,

which is considered to be rigid and the nodes connecting the tooth to the bone are considered fixed.

Application in Orthodontics and Dentofacial Orthopaedics

The use of FEM has been incorporated into dentistry and orthodontics to study the nature of stress and strain induced by orthodontic forces during tooth movement.[6] In 1983, Tanne et al.[3] applied finite element analysis to orthodontics research, thereby enabling a finite element model of a single structure, such as the temporomandibular joint[6], the nasal maxillary complex[7], mandible[8], etc. In most cases, biomechanical analysis using a single-structure model is neither accurate nor comprehensive.

With the advantage of being a non-invasive and accurate method that provides quantitative and detailed data on the physiological reactions possible to occur in tissues, applying the FEM can anticipate the visualization of these tissue responses through the observation of areas of stress created from applied orthodontic mechanics.[7]

The FEM principle is based on the division of a complex structure into smaller sections called elements in which physical properties, such as the modulus of elasticity, are applied to indicate the object response against an external stimulus such as an orthodontic force. It represents a great advantage of the method, since the degree of simplification can be controlled.[7]

Several studies on orthodontic force induced tooth movement were conducted using experimental animal models. These studies provide indications on the consequences of applying orthodontic forces to human tissues.[8] Since this type of experiment requires the use of living animals in laboratory, it is frequent that ethics committees on animal research have objections. With FEM, it is possible to anticipate the tissue responses to orthodontic mechanics applied. Alternative experimental models used to analyze the biomechanics of tooth movement include photoelastic models;[9] however, they have the disadvantage of exploring only the surface of the model, leaving internal structures, such as the periodontal ligament, behind.

For overcoming the aforementioned disadvantages, the FEM has reformed biomechanical research in Orthodontics. It represents a non-invasive, accurate method that provides quantitative and detailed data regarding the physiological responses occurring in tissues, such as the periodontal ligament and the alveolar bone.[10]

According to Middleton et al,[11] this accurate analysis of potential stress and tension occurring in tooth tissues is difficult to be obtained through any other experimental technique due to the interaction between surrounding tissues and the individual response.

CONCLUSION

FEM is a reliable experimental analysis that is easy, cost‑effective, and takes less time. Finite element analysis has proven itself an established numerical analysis with a paramount importance in not only dentistry and medicine, but also in orthodontics. The modelling and simulation steps save time and money for conducting the live experiment or clinical trial. Although finite element analysis is an accurate tool in assessing stress distribution, it is effective only for a given set of values or situation. However, situation and biomechanical properties of living structures could vary from person to person. Hence, the obvious shortcomings should be kept in mind before any decision-making procedure in experimental as well as clinical dentistry.

REFERENCES

1. Burnett, D. S. Finite element analysis: from concepts to applications. Addison-Wesley Pub. Co, 1987.
2. Reddy JN. An introduction to the finite element method. McGraw-Hill Higher Education, 2006.
3. The Finite Element Method in Engineering. Butterworth-Heinemann, 2011. at <http://www.myilibrary.com?id=96441>.
4. Lee NK, Baek SH. Stress and displacement between maxillary protraction with miniplates placed at the infrazygomatic crest and the lateral nasal wall: a dimensional finite element analysis. Am J Orthod Dentofacial Orthop, 2012; 141: 345-51.
5. Park JH, Bayome M, Zahrowski JJ, Kook YA. Displacement and stress distribution by different bone-borne palatal expanders with facemask: a 3-dimensional finite element analysis. Am J Orthod Dentofacial Orthop, 2017; 151: 367-74.
6. Cattaneo P M, Dalstra M, Melsen B. The finite element method: a tool to study orthodontic tooth movement. J Dent Res., 2005; 84: 428–433.
7. Knop L, Gandini Jr. LG, Shintcovsk RL, Gandini MREAS. Scientific use of the finite element method in Orthodontics. Dental Press J Orthod, 2015; 20(2): 119-25.
8. Jones ML, Hickman J, Middleton J, Knox J, Volp C. A validated finite element method study of orthodontic tooth movement in the human subject. Am J Orthod Dentofacial Orthop, 2001; 28(1): 29-38.
9. Caputo A, Chaconis SJ, Hayashi RK. Photoelastic visualization of orthodontic forces during canine retraction. Am J Orthod, 1974; 65(3): 250-9.
10. Kamble RH, Lohkare S, Hararey PV, Mundada RD. Stress distribution pattern in a root of maxillary central incisor having various root morphologies: a finite element study. Angle Orthod, 2012; 82(5): 799-805.
11. Middleton J, Jones M, Wilson A. The role of the periodontal ligament in bone modeling: the initial development of a time- dependent finite element model. Am J Orthod Dentofacial Orthod, 1996; 109(2): 155-62.