ISSN: 2581-5989
PubMed - National Library of Medicine - ID: 101738774

International Journal of Dental Science and Innovative Research (IJDSIR)

IJDSIR : Dental Publication Service

Available Online at: www.ijdsir.com

Volume – 5, Issue – 2, April - 2022, Page No. : 335 - 342

Finite element method in orthodontics (fem) - A review

[1]Dr. Divya Bharathi, Post Graduate Student Department of orthodontics and Dentofacial orthopedics Government college of Dentistry Indore, Madhya Pradesh India-452001

[2]Dr. Sandhya Jain, Professor and Head Department of orthodontics and Dentofacial orthopedics Government college of Dentistry Indore, Madhya Pradesh India-452001

[3]Dr. Nikita Choudhary, Post Graduate Student Department of orthodontics and Dentofacial orthopedics Government college of Dentistry Indore, Madhya Pradesh India-452001

Corresponding Author: Dr. Divya Bharathi, Post Graduate Student Department of orthodontics and Dentofacial orthopedics Government college of Dentistry Indore, Madhya Pradesh India-452001

Citation of this Article: Dr. Divya Bharathi, Dr. Sandhya Jain, Dr. Nikita Choudhary, "Finite element method in orthodontics (fem) - A review", IJDSIR- April - 2022, Vol. – 5, Issue - 2, P. No. 335 – 342.

Type of Publication: Review Article

Conflicts of Interest: Nil

Abstract

Engineering has not only developed in the field of medicine but has also become quite established in the field of dentistry, especially Orthodontics. Finite element analysis (FEA) was a computational procedure to calculate the stress in an element, which performs a model solution. This structural analysis allows the determination of stress resulting from external force, pressure, thermal change, and other factors. This method was 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 finite element method (FEM) to view a variety of parameters, and to fully identify implications of the analysis. It was extremely important to verify what the purpose of the study was in order to correctly apply FEM.

Keywords: Finite element in orthodontics, a review.

Introduction

The study of craniofacial orthodontics requires the precise understanding of the stress and strain induced by orthodontic forces in the periodontium.[1]

The teeth are moved to their ideal positions in the dental arch by means of orthodontic forces. But there are also mechanical considerations that need to be made since there is a stress-strain relation as well as force vectors involved. This is the reason why there needs to be a better understanding of the forces involved and the effects they bring about for which finite element analysis is becoming a very popular method.[2]

FEM in orthodontics is used to analyze the structural stress and is considered to be a highly precise technique.

An accurate model of tooth and its surrounding structures is offered by this method.

History

The history of Finite Element Analysis (FEA) dates back to 1940 for use in civil and aerospace engineering. Yet tram et al introduced this tool in orthodontics in 1972.[4]

In ,1943 when R. Courant first developed this technique. He utilized the Ritz method of numerical analysis and minimization of variation calculus to obtain approximate solutions to vibration systems.

Later in 1956, Turner MJ et al. published a paper thereby establishing broader definition of numerical analysis. The paper centered on the "stiffness and deflection of complex structures".

It was introduced in implant dentistry in 1976 by Weinstein.

Application of this technique in microcomputers, pre and post processors and for analysis of large structural system was in 1980's and 1990's.[5]

Important results obtained using finite element analysis

There are several studies that have already been conducted in the past using Finite element analysis.

One of the more important studies done using this method was by

1. McGuinness et al. who studied the forces brought about by edgewise appliances. They concluded that there was more stress near the cervical area of the tooth than at the root apex.
2. Another study was done by Kojima and Fukui who studied whether the teeth used for anchorage moved when a passive Trans-palatine bar was used. They found out that there was almost minimal effect when a mesializing force was applied.
3. A very relevant result was obtained by Tominaga et al. when they studied en-mass retraction using finite element method. They concluded that for controlled retraction of the anterior, the ideal position to place the hook is distal to the lateral incisor and mesial to the canine.[3-5]

Review of literature

FEM in Maxillary Protraction

Dong R et al established a three-dimensional finite element model of the craniofacial complex and analysed the force direction of protraction on the temperomandibular joint. The force applied on the chin was 5N. The angle of force direction was 40* relative to the occlusal plane when the stress and displacement was relatively small.

Liu c et al analyzed the maxillary growth based on the labiolingual appliance during the maxillary protraction treatment cycle. A three-dimensional stimulation of labiolingual appliance was performed.

The bio mechanical effects of maxillary protraction was also investigated by Chen zx et al with and without maxillary expansion on a model of unilateral cleft lip and palate before and after alveolar bone graft.[5]

FEM and Arch Wire Activation

Canales et al birth death technique in orthodontic brackets proved to be a bio mechanical stimulation for placement of continuous arch wires. On the application of retraction force the displacement of maxillary incisor and arch wire deformation were calculated.[6-10]

FEM and Alveolar Bone

3-D finite element model of maxillary first molars was established by Wang H et al and the stress magnitude and distribution within the Periodontal ligament of maxillary first molars was calculated when loaded with intrusion force.[7]

FEM and Mini -Implants

Holberg N et al found that when indirect mini_implant anchorage was used, FEM results revealed high loads on dental anchorage. Holberg C et al revealed that bio cortical implant anchorage is more favorable biomechanically than mono cortical anchorage.[6-10]

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 centered on the canine brackets.[9]

FEM and Trans Palatal Bar and Twin Block

Trans palatal arch design was used to optimize unilateral molar rotation correction using a finite element method by Gera my A et al. An activation of about 0.1 and 1.0 mm produced the same increasing patterns regarding the energy levels.[10]

FEM and Root Resorption

The initial effects of stress on the periodontal ligament were compared over time in orthodontic external root resorption, necrosis and TRAP + cell population.

FEM and Expanders

Araugio RM et al analysed that less dental tipping was generated when the ideal screw position is slightly above the maxillary first molar center of resistance.[6-10]

FEM in Orthognathic Surgery

The hybrid technique fixation of the Sagittal split ramus osteotomy was evaluated by Sato FR et al for their mechanical characteristics and stress distribution.

FEM in Brackets

The bracket design had less influence on the torquing moment. An increased torque control capability was exhibited by wider brackets. Holberg C et al attempted to create an anisotropic finite element method model of mandibular bone and orthodontic bracket. The results indicated that the risk of enamel fracture depended on the individual debonding procedure.[6-10]

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.

Methodology in finite element analysis

1. The most advanced and reliable study that revolutionized the dental and biomechanical research is the Finite Element Analysis/ Finite Element method (FEA/FEM) (FIG 1 & 2). A Cone Beam Computed Tomography (CBCT) of fully developed with maxillary and mandibular bone was taken (FIG: 3) CBCT output was taken as an STL file & was sent for processing This is a numerical form of analysis that allows stresses and displacements to be identified. (FIG 5,6 &7)

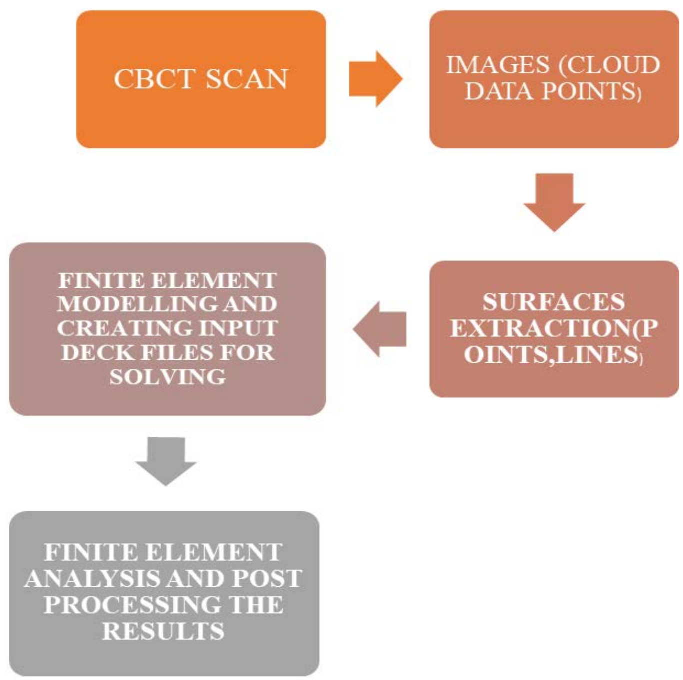

Fig 1: steps of construction of fem models

The object to be studied 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 called discretization, into a number of sub units termed elements. These are connected at a finite number of points called nodes (FIG 4). The results of FEM will be based upon the nature of the modeling systems and for that reason, the procedure for modeling is most important.[11]

Workflow of the Steps used in this study are as follows

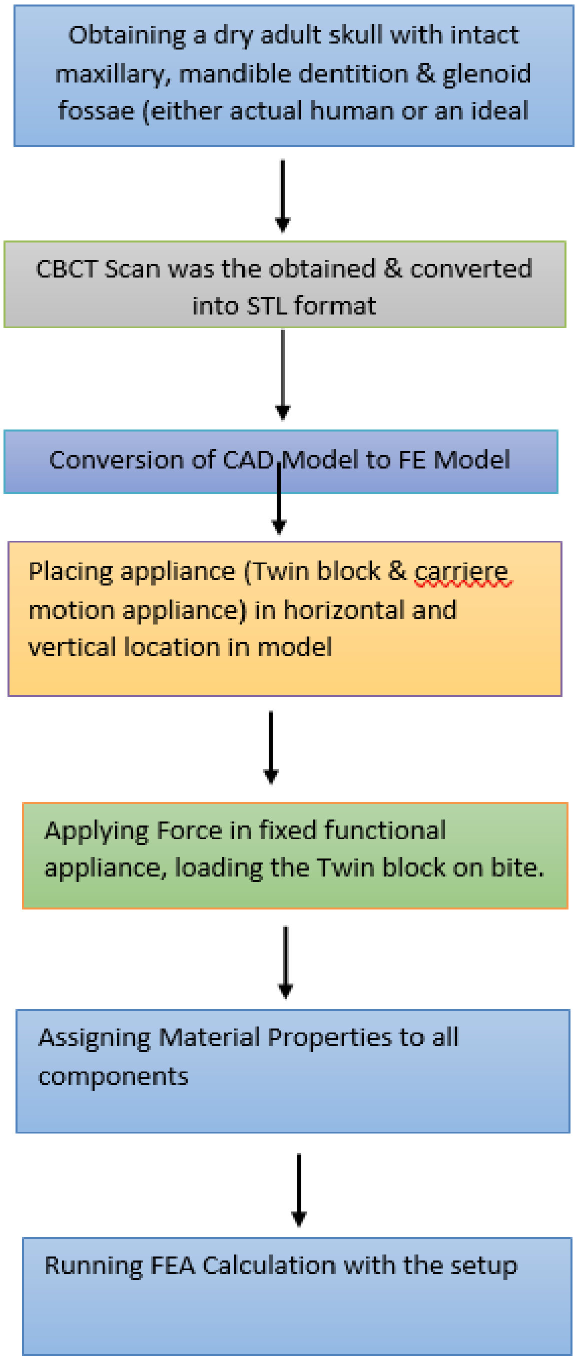

Fig-2: workflow of the steps used in this study.

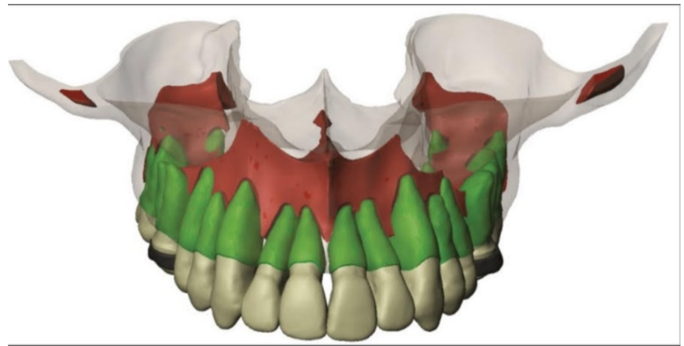

Fig 3: Virtual reconstruction of the maxilla by means of computed tomography.

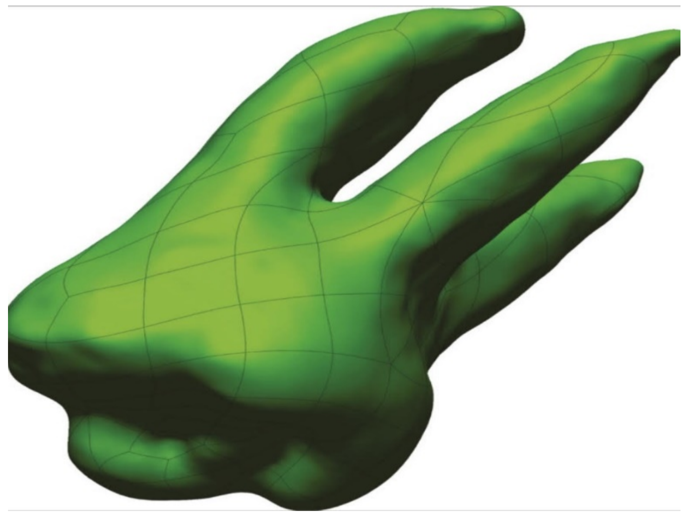

Fig 4: Example of elements and bonds of a molar. The bonds connect the elements to each other and are located in their extremities.

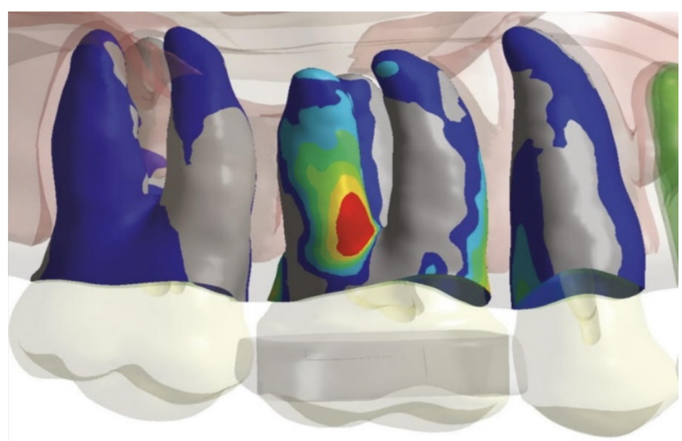

Fig 5: Areas of stress concentration on the periodontal ligament after submission to orthodontic force. Near the furcation area, we observe (red) higher stress concentration that gradually decreases towards the apex.

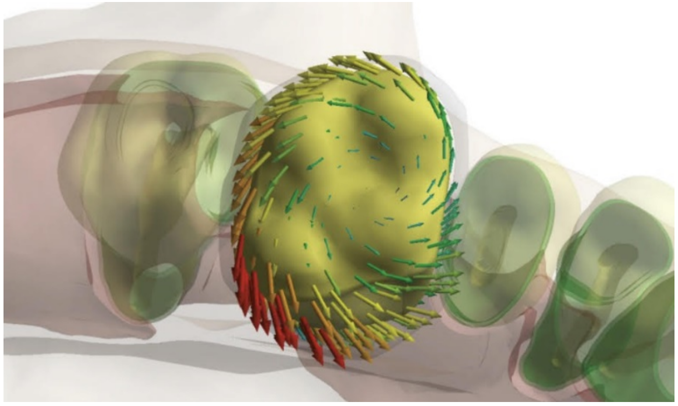

Fig 6: Arrows indicate the direction of tooth displacement and its intensity (red for higher displacement; green for lower displacement).

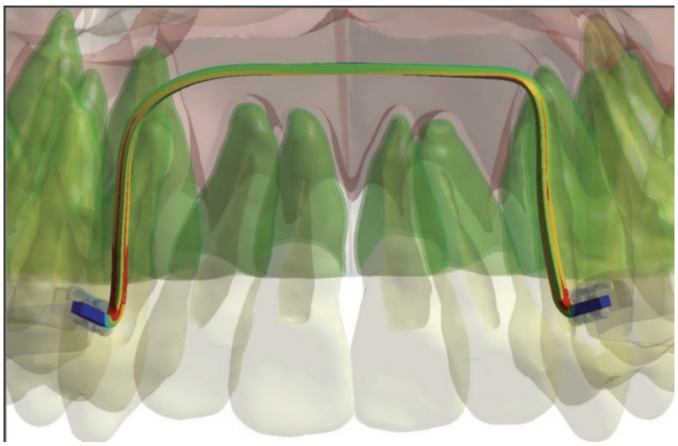

Fig 7: Areas of stress concentration on the orthodontic wire. Stress is more intense near the area with the wire bends (red).

Commercial FEM Software used,

- ABAQUS (Nonlinear and dynamic)
- ANSYS
- HYPER MESH (Pre/Post processor)

Basic steps involved in carrying out fea are

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

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 (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 3D – Laser scanner: Usually done for modelling inanimate objects.[11] For example modelling of brackets (FIG 3).

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.[12] The joining of elements at the nodes and eliminating duplicate nodes is termed as 'Meshing' (FIG 4).

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.

Sn.	Material	Young's modulus (GPA)	Poisons coefficient
1.	Acrylic	2.55	0.3
2.	Teeth	20.29	0.3
3.	Periodontal ligament	0.005	0.49

4.	Cortical bone	14.5	0.323
5.	Cancellous bone	1.37	0.3

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.[11]

Loading configuration

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

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.

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.

Application in orthodontics

Finite element modeling can be applied in three areas of biomechanics:

(i) analysis of the skeleton,

(ii) analysis and design of orthopedic devices and

(iii) analysis of tissue growth, remodelling and degeneration.

The FEM can also be applicable to the problem of stress strain levels induced in internal structures. This method also has the potential for equivalent mathematical modeling of a real object of complicated shape and different materials. Thus, FEM offers an ideal method for accurate modelling of the tooth and periodontium with its complicated 3- dimensional geometry.

The force systems used on an orthodontic patient are complicated. The FEM makes it possible to analytically apply various force systems at any point and in any direction. Orthodontic tooth movement is achieved by remodeling processes of the alveolar bone, which are triggered by changes in the stress/strain distribution in the periodontium. The finite element (FE) method can be used to describe the stressed situation within the periodontal ligament (PDL) and surrounding alveolar bone. It can also be used as a tool to Study Orthodontic Tooth Movement .[10-12]

FEM has been useful tool for morphometric analysis in craniofacial biology. The cephalometric finite element analysis (CEFEA) program incorporates the advanced features of the finite element method but bypasses the detailed understanding of the engineering and mathematics previously required to interpret results.

The program uses the color graphics display of common personal computers to show size change, shape change, and angle of maximum change. These are pictured as colored triangles of clinically relevant regions between pre- and mid- or post treatment lateral head films. The program is designed to have features of interest in both clinical practice and research .[12]

The effect of altering the geometry of the bracket base mesh on the quality of orthodontic attachment employing a three-dimensional finite element computer model is another application of FEM in orthodontics as discussed by Knox et al. The CAD/CAM template gives orthodontists safe way to place Minis crews.

Advantages of fem

FEM can be applicable to linear and non-linear as well as solid and fluid structural interactions.

- Any problems can be split into smaller number of Problems.
- It is a non-invasive technique.
- By using FEA, it's very easy to simulate any biological condition in pre, intra and post-operative stages to achieve more accurate and reliable results.
- Reproducibility does not affect the physical properties involved.
- FEA techniques can replace stereo lithographic models for presurgical planning, thus provides an economical solution for the same.
- Static and dynamic analysis can be done.
- This technique is 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.
- It does not require extensive instrumentation.
- The study can be repeated as many times as the operator wants

Drawbacks

The roots, periodontal ligament and teeth are represented in idealized geometric forms and physical properties are assumed to be homogenous, isotropic and linear. Various different methods which can also be applied are laser holography and photo elastic stress analysis.[11]

Conclusion

FEM is a reliable experimental analysis that is easy, cost-effective, and takes less time. This field has not only taken orthodontics to new heights but is also used extensively by other fields of dentistry and medicine. The role of FEM in treatment planning, bone remodeling, determining the center of resistance and rotation, and retraction has helped in understanding the biomechanics of tooth movement. Further, newer ideas can be easily implied using FEM

References

1. Moss ML. The Application of The Finite Element Method to The Analysis of Craniofacial Growth and Form. Acta Morphol Neerl Scand. 1985; 23:337-5
2. Moss ML, Et Al. Finite Element Method of Modeling of Craniofacial Growth. AR, I J Orthod 1985;87:453-71.
3. Vollmer D, Meyer U, Joos U, Vegh A, Piffko J. Experimental and Finite Element Study of a Human Mandible. J Craniomaxillofac Surg 2000; 28:91-
4. Beek M, Koolstra JH, Van Ruijven LJ, Van Eijden TM. Three-Dimensional Finite Element Analysis of The Human Temporomandibular Joint Disc. J Bio mech. 2000;33(3):307–16.
5. Vollmer D, Meyer U, Joos U, Vegh A, Piffko J. Experimental and Finite Element Study of a Human Mandible. J Craniomaxillofac Surg. 2000;28(2):91–6.
6. Mackerle J. Finite Element Modelling and Simulations in Dentistry: A Bibliography 1990–2003. Computer Methods in Biomechanics and Biomedical Engineering. 2004 Oct 1;7(5):277-303.
7. Srirekha A, Kusum Bashetty. Infinite To Finite: An Overview of Finite Element Analysis. Indian Journal of Dental Research: Official Publication of Indian Society for Dental Research. July 2010, 21(3):425-32

8. Prasad Konda, Tarannum SA. Basic Principles of Finite Element Method and Its Applications in Orthodontics. Issn No- 2230 – 7885/ Jpbms, 2012, 16 (11).
9. Shilpa Trivedi. Finite Element Analysis: A Boon to Dentistry. Journal of Oral Biology and Craniofacial Research.2014 200-203
10. Mohammed SD, Desai H (2014) Basic Concepts of Finite Element Analysis and Its Applications in Dentistry: An Overview. Oral Hyg Health 2: 156. Doi: 10.4172/2332-0702.1000156
11. Knop L, Gandini LG, Shintcovsk RL, Gandini MR. Scientific Use of The Finite Element Method in Orthodontics. Dental Press Journal of Orthodontics. 2015 Mar; 20:119-25.
12. Marya A, David G, Eugenio MA. Finite Element Analysis and Its Role in Orthodontics. Adv Dent & Oral Health. 2016 Jul 28; 2:5-6.