An Overview of Basic Concepts of Finite Element Analysis and Its Applications in Orthodontics

Shafagh Rastegari[1], Seyed Majid Hosseini[2], Mojtaba Hasani[3] and Abdolreza Jamilian[4,5,*]

[1]*Department of Computer Engineering, Iran University of Science and Technology, Tehran, Iran*

[2]*Department of biomedical engineering, Politecnico di Milano, Italy*

[3]*Mechanical Engineering Department, Iran University of Science and Technology, Tehran, Iran*

[4]*Module leader, City of London Dental School, University of Bolton, London, UK*

[5]*Orthodontic Department, Faculty of Dentistry, Tehran Medical Sciences, Islamic Azad University, Tehran, Iran*

Abstract: *Purpose:* The aim of this article is to acquaint the readers with the aims and goals of the finite element method and how to use it in dentistry and especially in orthodontics.

Methods: The finite element method (FEM) has shown to be a beneficial research tool that has assisted scientists in various analyses such as stress-strain, heat transfer, dynamic, collision, and deformation analyses. The FEM is responsible for predicting the behavior of objects under different working conditions. It is a computational procedure to measure the stress in an element, which performs a model solution to solve a problem; the FEM subdivides a large system into smaller, simpler parts called finite elements. This is achieved by a particular space discretization in the space dimensions, which is implemented by the construction of a mesh of the object. The technique of FEA lies in the development of a suitable mesh arrangement.

Conclusions: The FEM can be effective in understanding the behavior of teeth, both jaws, craniofacial structure, and other hard tissue structures of humans under various working conditions, as the technique allows for evaluating tooth movement and the stress distribution within the surrounding alveolar bone, the periodontal ligament (PDL). This technique is exceptionally valuable for evaluating mechanical aspects of biomaterials and human tissues that can hardly be measured *in vivo*. This review article presents the FEM, its methodology, and its application in the orthodontic domain.

Keywords: Finite element analysis, Finite element method, Orthodontics.

INTRODUCTION

Experimental, analytical, and numerical methods are well-known techniques used to solve problems [1]. Each method has some advantages and disadvantages making them practical or impractical in different problems. For instance, although experimental technique can lead to trustable outcomes, it has some drawbacks such as being time consuming, can be influenced from human error, being high-cost technique, all variables cannot be controlled, and its result can be applied to only one situation [2]. Furthermore, employing analytical methods would be really challenging and, in some cases, inapplicable for complex problems (like alveolar structures). Therefore, the numerical method can be a good choice in terms of time, cost, eliminating human error, managing the variables, repeating the analyses, and solving complex problems. The Finite Element Method (FEM) is regarded as a numerical method that can be employed to model physical or physiological phenomena. Computers using the FEM, simulate different physical and geometrical characteristics of any structures. This method can assist in the prediction of behavior of any object under differing situations, and can help with the design and manufacturing process [3].

Finite element analysis (FEA) evaluates physical phenomena using a numerical technique, the FEM. It is a powerful computerized method that can analyze the behavior of complex geometry with heterogeneous material properties. FEA is employed in orthodontics by evaluating the interactions between the teeth and the surrounding tissues. The method allows orthodontists to understand better the physiological reactions that are likely to occur within the dentoalveolar complex, resulting from thermal stress created by hot and cold drinks [4].

FEA was first developed by R. Courant [5] in 1943 using the Ritz method in numerical analysis. A few years later, in 1956, Turner MJ *et al*. [6] published a paper focusing on the "stiffness and deflection of

*Address correspondence to this author at the Module leader, City of London Dental School, University of Bolton, London, UK. Orthodontic Department, Faculty of Dentistry, Tehran Medical Sciences, Islamic Azad University, Tehran, Iran; Tel: 0098-22052228; E-mail: info@jamilian.net

complex structures. FEA was introduced in dentistry in 1976 and Application of this technique in microcomputers, pre and post processors and for analysis of large structural system was in 1980's and 1990's [7]. The FE method was employed in orthodontic domain from 1973 and since then it has been used in many stress-strain analysis of alveolar structures [8]. Moreover, this method has been employed in validation of theories in dental biomechanical research such as theories describing alveolar tissue reactions associated with tooth movement [9, 10].

The aim of this paper is to give a general overview of FEA, and to describe its application for situations that cannot be reproduced in humans because of ethical considerations, or the difficulty of conducting research projects. This paper also illustrates the advantages and disadvantages of FEA.

METHODOLOGY IN FINITE ELEMENT ANALYSIS

FEM divides a complex structure (like individual teeth or the entire mandible) into a system of smaller elements or points (nodes) using the 'mesh generation technique'. Mesh generation is subdivision of a geometric space into small, simple shapes such as triangles or quadrilaterals, in two dimensional cases and pyramids, prisms, hexahedra, or tetrahedra in structures where the thickness of objects cannot be ignored, (see Figure **1**).

After defining the problem (like predicting physiological reactions that are likely to occur in teeth and the mandible, under orthodontics treatment) using FEM, the calculations needed to obtain results are first performed on the nodes. In order to obtain stress distribution on the wire in orthodontic treatment for example, on each node within the elements, the stress value can be calculated using Hooke's law (F=KX). Knowing the stiffness for each element, the stress value can be obtained by multiplying displacement of element determined by software and the stiffness of element. The model is created based on the structure's shape in reality or some images like CT scan [11].

FEA as a computerized analysis allows the estimation of the biomechanical response. When defined forces and/or deformations are applied to the system, the biomechanical reaction of system would be obtained (*e.g.*, tooth movement as a result of applying force on the wire in orthodontic treatment). This method enables dental professionals to simulate and analyze dental structures. It allows them to benefit from a more reputable and harmless technique that replaces many unsafe and multifaceted procedures [12].

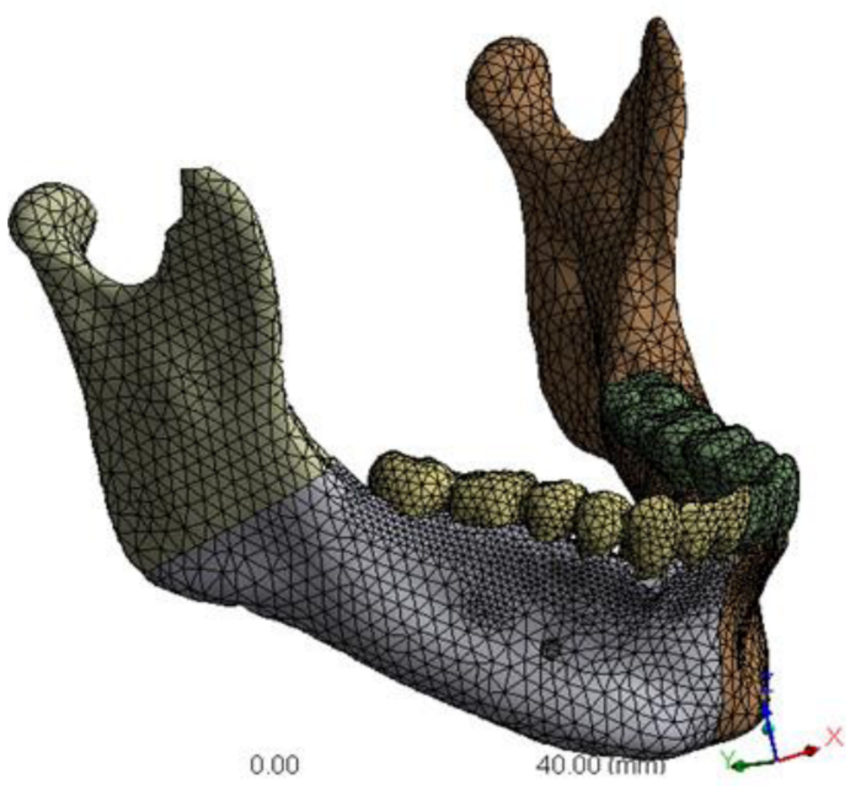

Figure 1: Meshed dental model on mandible.

BASIC STEPS INVOLVED IN CARRYING OUT FEA

There are some steps used in any FEA which are:

• Pre-Processing

a) Conversion of the structure into a finite element model

b) Assembly of the model and defining material property (such as density, elastic modulus, poisson ratio, yield stress, fracture toughness, specific heat)

c) Defining the boundary conditions (constrains which are applied on the real problem)

d) Applying the loads

e) Meshing the model (in which the geometry is divided into non-overlapping, small which are connected to the *nodes*)

• Processing

In this step, the computer software uses FEM to calculate the results, considering the defined situations.

• Post-Processing

After solving the problem, the obtained results would be viewed and compared with a real case. Since in some cases the difference between real case and FEM model cannot be ignored, therefore the model should be improved in terms of mesh, or defined boundary conditions, or material properties to produce results more similar to the real situation. A simulation will be developed to improve the accuracy of the results, making them more reliable.

The following sections will explain each step in more detail [13].

This process can be performed using Commercial FEM Software Packages like ABAQUS, ANSYS, NASTRAN, COMSOL.

Preprocessing

a) Conversion of Geometric model to Finite Element Model

The geometry, material properties, loading conditions, and boundary conditions applied on the simulated model should be copied from the boundary condition and loads in reality to obtain more meaningful and realistic results. For this purpose, to have a more accurately simulated model in terms of geometry of the model, the geometry of the structure can be achieved using different methods such as:

- Laser Scan: Usually used for modeling inanimate objects, *e.g*., brackets.

- Computerized tomography (CT) scan: Usually used for complex structures or living tissues, *e.g*., craniofacial skeleton, mandible, and maxilla.

Regarding the images or scans of the natural structure, the model can be generated using points, lines, areas, and volumes [14].

b) Assembly / Material Property Data Representation

Each component can be modeled separately and assembled to create the entire structure, *e.g*., each tooth can be created separately and then will be put in its location in jaw to create the entire structure (see Figures **1** and **2**). Subsequently, the material properties will be assigned to the system, bearing in mind that the minimum requirement for each material is Young's modulus and Poisson's ratio. In this section, for each of the components, the software allows us to assign its special material properties and these material properties are obtained from experimental work. Table **1** presents the Young's modulus and Poisson's ratio for some biomaterials [15].

Table 1: Material Properties

Material	Young's modulus (MPa)	Poisson's ratio
Tooth	20000	0.3
PDL	0.05	0.3
Alveolar bone	2000	0.3
Composite	14200	0.3
Bracket	200000	0.3
Arch wire	200000	0.3

c) Defining the Boundary Conditions

The next step is applying boundary conditions at the boundary of the simulated model to achieve more reliable results. These conditions are applied to the model considering the real case's situation. No translation or rotation (in each direction x, y, and z) should be experienced in this case. These boundary condition can be applied to the nodes, lines, surfaces, and bodies of the simulated model based on constraints imposed on the real case of study [12].

d) Loading Configuration

In FEA, loads, forces, deformations can be applied on nodes, action lines, areas, or bodies in any direction. The loads can be of different types such as mechanical, thermal, or electrical. For instance, in dentistry, the stress distribution in a tooth, after collision of a hard object with that tooth, can be simulated by imposing a load of a predescribed value in the intended direction.

e) Meshing the Model

The process in which the geometry is divided into small elements is called discretization. These elements should not be overlapped and are only connected to the *nodes*. This system of nodes and elements makes a grid that is called mesh. As shown in Figure **1** and mentioned above, the mesh can be triangles or quadrilaterals in 2D structures, and pyramids, prisms, hexahedra, or tetrahedra in 3D structures. Gathering these elements (meshes) together will create the entire structure that will be analyzed using FEM, and there should not be any gaps or overlaps.

In order to analyze a structure, such as performing a thermal stress analysis of teeth when a hot drink is contacting the teeth, the thermal stress should be calculated in whole model. For this purpose, the numerical equations are solved for each node and element in the mesh and are later assembled to generate the global equation that presents the properties of the whole structure [11]. Meshed dental models on the mandible and the maxilla can be seen in Figures **1** and **2** respectively.

Processing

A set of global equations, regarding the load and boundary conditions for the whole system is produced, and the desired output like stress and strain are obtained employing FEA. In this step, material properties, load and boundary conditions, and acceptable mesh size considering the size of the system can influence the results significantly (7).

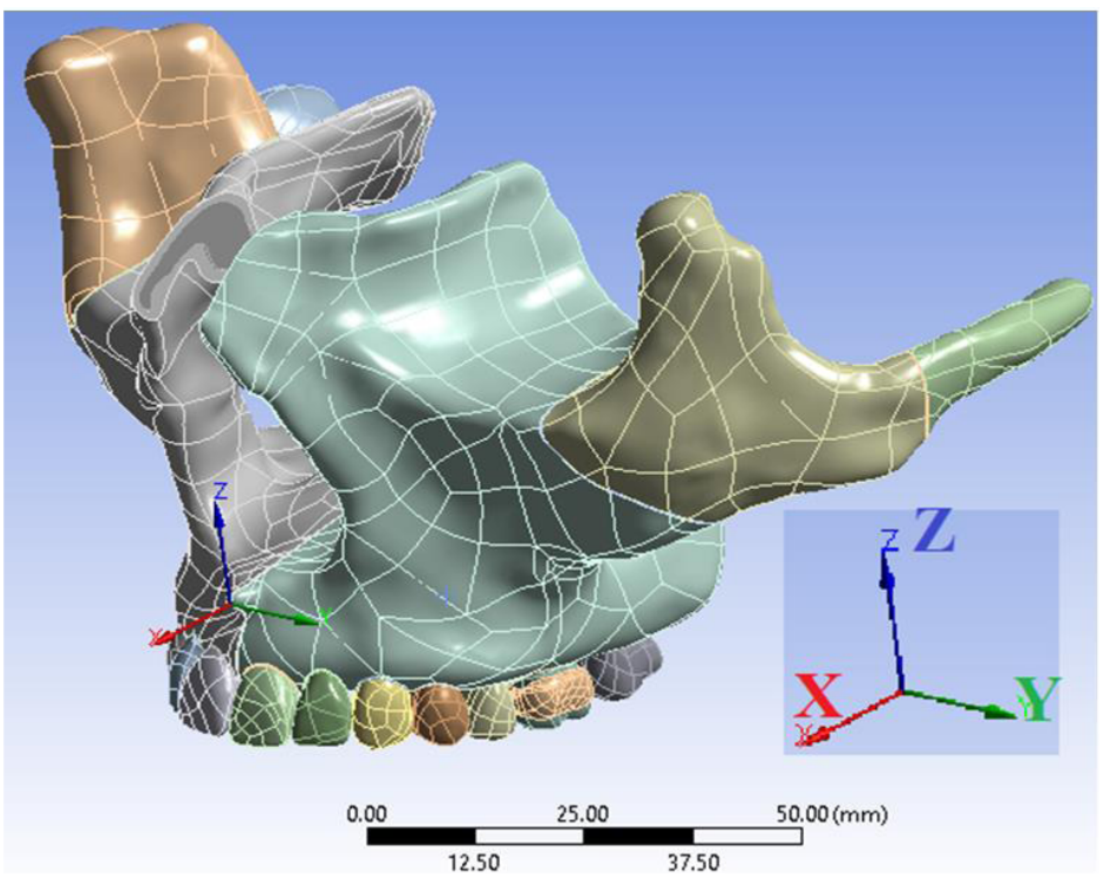

Figure 2: Meshed dental model on maxilla.

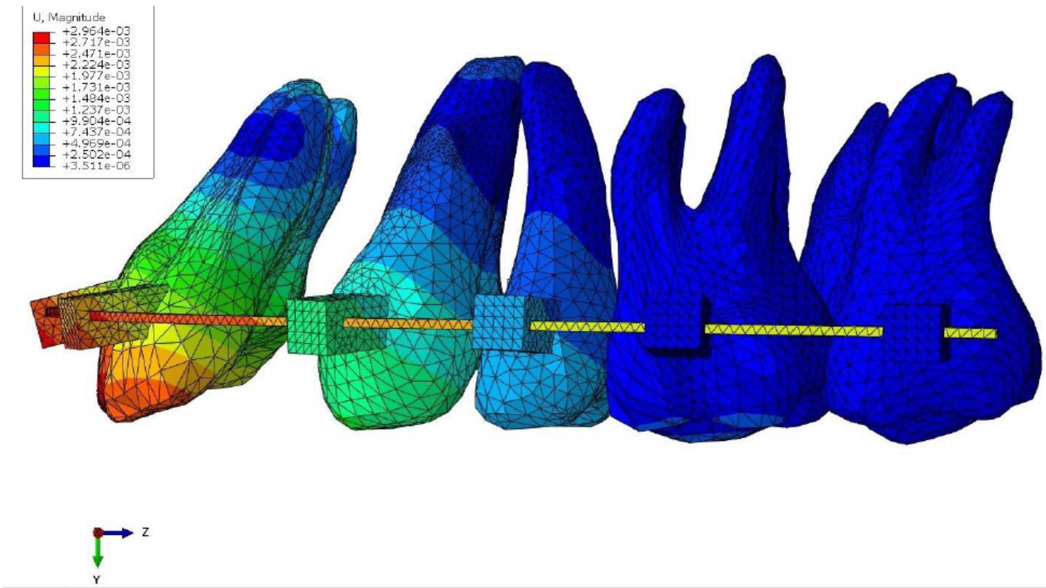

Figure 3: Initial tooth movement patterns utilizing the high stiffness arch wire.

Post-Processing

After calculating the intended outputs on each node and element by FE analysis, they can be presented as graphical output (like deformation of teeth resulting from some forces, as is illustrated in Figure **3**). The data obtained from FEM can be represented as the user wants. For example, if user wants the deformation distribution for teeth, deformation can be selected from menu bar of software.

DISCUSSION

As mentioned above, FEM is a powerful technique that can help scientists to solve complex problems by dividing the continuous system domain into elements and altering the problem from an infinite domain to small finite elements that are smaller and easier to solve. The required variables are calculated in nodal points of each element utilizing interpolation and then by having values in every single node, the variable for whole structure would be obtained. In other words, the intended outputs are later obtained by assembling the nodal results and generating the entire system matrix function. [16] Even though FEA is a widespread technique with many advantages, some disadvantages are also present.

Advantages of Finite Element Method

- Simulation of irregular geometry with high complexity and using images obtained by scans of geometry helps in accuracy.
- Easy to handle almost all kinds of boundary conditions and loads.
- Assigning different types of materials properties (including anisotropic and nonhomogeneous) to each component of the system
- Appliable for both linear and nonlinear and static and dynamic problems
- Easy to use and non-invasive.
- Infinitely repeatable to assess the impact of changing conditions on the results.
- Visualize the results and help scientists understand the critical points and see the intended output. For example, presenting the maximum stress point helps scientists with possible failure points.
- Help reduce laboratory testing, which can be significant time and cost effective. [10]

Disadvantages of Finite Element Method

- A large quantity of data is required, which should be very precise. Otherwise, the results would not be reliable.
- Some assumptions can lead to errors in the results. However, they must be made (*e.g.*, assuming rigid teeth in surrounding tissue which is not realistic.)

- In some cases, modeling biological structures is difficult considering the complexity of the anatomy. Hence, some idealization in terms of shape and mechanical properties is required, affecting the accuracy of the results [11].

Applications of FEM

There is a wide range of applications for FEM in different areas: Automobile, Aerospace, Civil Engineering, Geo Mechanics, Orthopedic research, Biomathematics and Biomechanics. One of the applications of FEM, which can significantly impact human life, is the application of this method in *biomedical engineering,* which can help reduce the required cost and time and increase the quality of life [17].

FEM in Biomedical Engineering

The first use of FEM in the biomedical field goes back to 1972 when FEM was employed to perform stress analysis in the orthopedic domain [18]. This method provides reliable results and is beneficial in terms of timesaving and cost-effectiveness. FEM's popularity has grown sharply in all biomedical areas. Besides, three domains of biomechanics in which application of FEM is expected include (i) skeleton stress or failure analysis, (ii) orthopedic devices' design, and (iii) tissue engineering. This method can be used for internal structures subjected to different loads and stresses under different boundary conditions. FEM is a perfect method to simulate orthodontic issues, especially the teeth and their periodontium. This is because of FEM's ability to model complex shapes such as teeth and mandible shapes and assign different materials to each component [18].

a) Application in Orthodontics

In the simulation of the orthodontic domain using FEM, applying various load systems at any node or area in any direction is possible, making this method more comprehensive and popular.

This mathematical method helps us evaluate the human mandible in different situations under physiological conditions. It also responds to the Jawbone under forces employing 3D simulation. This is helpful in the design and selecting the most efficient location of miniplate osteosynthesis [19]. Furthermore, by employing FEM, many studies like tooth movement governed by changing stress and strain distribution in periodontium tissue, the stress distribution within surrounding alveolar bone and the periodontal ligament (PDL), orthodontic implants have been carried out:

Mestrovic *et al*. [20] reported that the more gingivally the applied force is, the bigger the movement of the tipping tooth would be. This was examined after studying the tooth movement from orthodontic forces.

Gallas *et al*. [21]. Simulated 3D bone and implant using FEM and found out the concentration of stress usually would be around the cervical region.

Moreover, Ghosh *et al*. performed stress analysis for six ceramic brackets using FEM. They discovered that the concentration of stress would be at edges, corners, and places there is a sharp change in the shape of the bracket.

Knox *et al*. [22]. studied the changes in stress distribution in the bracket-cement-tooth system by modifying the physical and geometrical features of cement. They presented that cement lute thickness and shape of the cement lute periphery can influence the stress distribution in the bracket-cement-tooth system. Moreover, Williams and Edmundson [23] utilized FEM in their analysis. They reported that the instantaneous center of rotation (ICR) of a maxillary central incisor does not depend on the periodontal ligament's elastic properties. At the same time, its position is related to the loading point. Furthermore, using the computer-aided design (CAD)/ Computer-aided manufacturing (CAM) template helps orthodontists to place mini-screws safely [24].

FEM can also be used in the prosthodontics domain and help design fiber framework for fixed partial dentures. In addition, it can be employed in Periodontics where the stress distribution in periodontium is evaluated according to the geometry and dimensions of dental implants [25].

CONCLUSION

In conclusion, FEM is employed in numerous fields as a powerful cost and time-effective technique. One area that has gained significant benefit from FEM is the orthodontic domain. In general, FEM divides a complex structure into a system of smaller elements (finite element model), subsequently, after assigning material properties and defining the boundary conditions for the model, the loads would be applied on the simulated model. In the next steps, meshed model (structure which is subdivided into small and simple shape elements) would be analyzed by calculating intended variable in each element, consequently, the outcomes would be visualized in software using FEM. Even though biological structures are very complex to be simulated because of their irregular shape and complicated material properties like viscoelastic properties, FEM can effectively calculate stresses and strains resulting from forces, design of implants and brackets, and determining periodontal stress and tooth movement. FEM therefore has an important role in applying new ideas and evaluating their efficiency in orthodontic domain.

ETHICS APPROVAL AND CONSENT TO PARTICIPATE

This is an overview, and there is no need for Ethical approval.

COMPETING INTERESTS

The Authors declare that they have no financial or non-financial competing interests.

FUNDING

There is no funding.

REFERENCES

[1] Tajima, K., *et al*., Three-dimensional finite element modeling from CT images of tooth and its validation. Dental materials journal, 2009. 28(2): p. 219-226. https://doi.org/10.4012/dmj.28.219

[2] Brekelmans, W., H. Poort, and T. Slooff, A new method to analyse the mechanical behaviour of skeletal parts. Acta Orthopaedica Scandinavica, 1972. 43(5): p. 301-317. https://doi.org/10.3109/17453677208998949

[3] Srirekha, A. and K. Bashetty, Infinite to finite: an overview of finite element analysis. Indian J Dent Res, 2010. 21(3): p. 425-32. https://doi.org/10.4103/0970-9290.70813

[4] Reddy, M.S., R. Sundram, and H.A. Eid Abdemagyd, Application of Finite Element Model in Implant Dentistry: A Systematic Review. J Pharm Bioallied Sci, 2019. 11(Suppl 2): p. S85-S91. https://doi.org/10.4103/JPBS.JPBS_296_18

[5] Courant, R., Variational methods for the solution of problems of equilibrium and vibrations. Bull. Amer. Math. Soc., 1943. 49: p. 1-23. https://doi.org/10.1090/S0002-9904-1943-07818-4

[6] Turner, M.J., *et al*., Stiffness and deflection analysis of complex structures. journal of the Aeronautical Sciences, 1956. 23(9): p. 805-823. https://doi.org/10.2514/8.3664

[7] Prasad K, T.S., Basic principles of finite element method and its applications in orthodontics. Journal of Pharmaceutical and Biomedical Sciences (JPBMS), 2012. 16(16): p. 1-8.

[8] Farah, J., R.G. Craig, and D.L. Sikarskie, Photoelastic and finite element stress analysis of a restored axisymmetric first molar. Journal of biomechanics, 1973. 6(5): p. 511-520. https://doi.org/10.1016/0021-9290(73)90009-2

[9] Cattaneo, P., M. Dalstra, and B. Melsen, The finite element method: a tool to study orthodontic tooth movement. Journal of dental research, 2005. 84(5): p. 428-433. https://doi.org/10.1177/154405910508400506

[10] Toms, S.R. and A.W. Eberhardt, A nonlinear finite element analysis of the periodontal ligament under orthodontic tooth loading. American Journal of Orthodontics and Dentofacial Orthopedics, 2003. 123(6): p. 657-665. https://doi.org/10.1016/S0889-5406(03)00164-1

[11] Reddy, M.S., R. Sundram, and H.A.E. Abdemagyd, Application of finite element model in implant dentistry: a systematic review. Journal of pharmacy & bioallied sciences, 2019. 11(Suppl 2): p. S85. https://doi.org/10.4103/JPBS.JPBS_296_18

[12] Singh, J.R., *et al*., Revolution in Orthodontics: Finite element analysis. J Int Soc Prev Community Dent, 2016. 6(2): p. 110-4. https://doi.org/10.4103/2231-0762.178743

[13] Kupczik, K., Virtual biomechanics: basic concepts and technical aspects of finite element analysis in vertebrate morphology. J Anthropol Sci, 2008. 86: p. 193-8.

[14] Cattaneo, P.M. and M.A. Cornelis, Orthodontic Tooth Movement Studied by Finite Element Analysis: an Update. What Can We Learn from These Simulations? Curr Osteoporos Rep, 2021. 19(2): p. 175-181. https://doi.org/10.1007/s11914-021-00664-0

[15] Geng, J.P., K.B. Tan, and G.R. Liu, Application of finite element analysis in implant dentistry: a review of the literature. J Prosthet Dent, 2001. 85(6): p. 585-98. https://doi.org/10.1067/mpr.2001.115251

[16] Cozzani, M., *et al*., The effect of Alexander, Gianelly, Roth, and MBT bracket systems on anterior retraction: a 3-dimensional finite element study. Clin Oral Investig, 2020. 24(3): p. 1351-1357. https://doi.org/10.1007/s00784-019-03016-6

[17] Guy, A., *et al*., Braces Designed Using CAD/CAM Combined or Not With Finite Element Modeling Lead to Effective Treatment and Quality of Life After 2 Years: A Randomized Controlled Trial. Spine (Phila Pa 1976), 2021. 46(1): p. 9-16. https://doi.org/10.1097/BRS.0000000000003705

[18] Huiskes, R. and E.Y. Chao, A survey of finite element analysis in orthopedic biomechanics: the first decade. J Biomech, 1983. 16(6): p. 385-409. https://doi.org/10.1016/0021-9290(83)90072-6

[19] Huang, Y., *et al*., Numeric modeling of torque capabilities of self-ligating and conventional brackets. Am J Orthod Dentofacial Orthop, 2009. 136(5): p. 638-43. https://doi.org/10.1016/j.ajodo.2009.04.018

[20] Mestrovic, S., M. Slaj, and P. Rajic, Finite element method analysis of the tooth movement induced by orthodontic forces. Coll Antropol, 2003. 27 Suppl 2: p. 17-21.

[21] Gallas, M.M., *et al*., Three-dimensional numerical simulation of dental implants as orthodontic anchorage. Eur J Orthod, 2005. 27(1): p. 12-6. https://doi.org/10.1093/ejo/cjh066

[22] Knox, J., *et al*., An evaluation of the influence of orthodontic adhesive on the stresses generated in a bonded bracket finite element model. Am J Orthod Dentofacial Orthop, 2001. 119(1): p. 43-53. https://doi.org/10.1067/mod.2001.110987

[23] Williams, K.R. and J.T. Edmundson, Orthodontic tooth movement analysed by the Finite Element Method. Biomaterials, 1984. 5(6): p. 347-51. https://doi.org/10.1016/0142-9612(84)90033-4

[24] Liu, H., *et al*., Accuracy of surgical positioning of orthodontic miniscrews with a computer-aided design and manufacturing template. Am J Orthod Dentofacial Orthop, 2010. 137(6): p.

728 e1-728 e10; discussion 728-9. https://doi.org/10.1016/j.ajodo.2009.12.025

[25] Anitua, E., *et al*., Influence of implant length, diameter, and geometry on stress distribution: a finite element analysis. Int J Periodontics Restorative Dent, 2010. 30(1): p. 89-95.

Received on 03-06-2023 Accepted on 03-07-2023 Published on 05-07-2023

DOI: https://doi.org/10.12974/2311-8695.2023.11.04