Section A-Research paper
ISSN 2063-5346

FINITE ELEMENT ANALYSIS OF ORTHODONTIC FORCES ON DIFFERENT ORTHODONTIC MATERIALS AND SURROUNDING DENTOFACIAL STRUCTURES

1.Dr.Mohamed Parvaize, 2. Dr. Vivek Amin, 3. Dr. Mohammed Safvan M A, 4. Dr.Mohammed Ashraf, 5. Dr Minaz, 6. Dr.Shahir Backer

[1]Post graduate, Orthodontics and Dentofacial Orthopedics, Yenepoya Dental College, Yenepoya(Deemed to be University), Mangalore
mohammedparvaize@gmail.com

[2]Professor, Department of Orthodontics and Dentofacial Orthopedics, Yenepoya Dental College, Yenepoya (Deemed to be University), Mangalore

[3]Postgraduate, Department of Orthodontics and Dentofacial Orthopedics, Yenepoya Dental College, Yenepoya (Deemed to be University), Mangalore

[4]Department of Orthodontics and Dentofacial Orthopedics, Yenepoya Dental College, Yenepoya (Deemed to be University),Mangalore

[5]Senior Lecturer, Department of Orthodontics and Dentofacial Orthopaedic, Yenepoya Dental College, Yenepoya (Deemed to be University),Mangalore

[6]Consultant Orthodontist, Carewell Dental Clinic, Kannur, Kerala

Corresponding author

Dr.Mohamed Parvaize, Post graduate, Orthodontics and Dentofacial Orthopedics, Yenepoya Dental College, Yenepoya(Deemed to be University), Mangalore
mohammedparvaize@gmail.com

DOI: 10.48047/ecb/2023.12.si4.1730

Abstract:

Objective: To evaluate the effect of varying orthodontic forces on different orthodontic materials and surrounding dentofacial structures using Finite Element analysis. **Materials and methods**: CT scan of the patient is converted into digital images with the help of MIMICS 14.0 software. Dimensions of mini-implant screws were obtained using scanned electron microscopy. Mini impalnts, and archwires are constructed using CATIA V5 R19 software. Blue light scanner is used for scanning brackets. Maxillary dentition including, PDL, and surrounding Bone were modelled using MIMICS software's. Brackets, mini-implant screws and archwires are modelled using CATIA software. Meshing of all the models is carried out using ANSYS software. **Results:** The maximum and minimum stress with different orthodontic materials in the mini-implant screws, surrounding bone, dentition, and PDL were recorded. ANSYS 15.0 software is used to carry out Finite Element Analysis. After applying Loads and Material Properties, solving is done. **Conclusion**: Application of varying orthodontic force produces maximum stress distribution in various dentofacial structures. While increasing the frequency of forces, there is significant decrease in stress distribution

Keywords: Finite element model, ANSYS, mesh model, orthodontic forces.

Introduction: Despite the orthodontist's extensive clinical expertise and outstanding diagnostic judgement, patients may be reluctant to seek treatment because of the length of the

ISSN 2063-5346

procedure and the associated discomfort, which are the two main concerns before deciding to undergo a procedure. Depending on the severity of the situation, orthodontic therapy lasts an average of two years or longer. The number of appointments needed tends to expand dramatically over time, lasting up to 32 months, depending on a variety of conditions, including the requirement for surgical treatment and patient compliance.[1] In response to an external force, the alveolar bone is modelled and remodelled during orthodontic tooth movement (OTM).[2]

The use of the FEM (finite element method), a well-established predictive technique, has gained popularity in biomedical research over the past few decades. It can simulate shape, analyze and calculate stresses and displacement in dentomaxillofacial structures,[3, 4] and create an abstraction of physical reality that can be studied for challenging clinical procedures.

It is a non-invasive, precise procedure that offers quantitative and in-depth information about the physiological reactions to orthodontic mechanics used to treat various types of malocclusion occurring in tissues, such as the periodontal ligament (PDL) and the alveolar bone[3.]

The Finite Element Method is a remarkably accurate method for analyzing structural stress because it breaks down extremely complex issues into manageable pieces. The FEM demands that the entire geometry, including the surrounding area, be simulated with finite elements for a specific design. It can be used to investigate various constructions and simulate varied stresses and material qualities.[5]

The finite element method (FEM) is an approximation technique that separates the total area of the structure into a collection of elements linked by nodes. To represent the physical attributes of the model, element types are chosen, and material properties are applied to each element. To simulate applied loads and constraint of the structure, the forces and boundary conditions are defined. The architectural response is computed and presented for display. The deformation of each element must be compatible with other elements and is determined by the equilibrium condition of the entire structure.[5]

AL Yettram et al. brought finite element analysis to orthodontics (1977). Compared to other stress analysis techniques, the FEM offers certain clear advantages. It is able to describe structures with irregular geometry and non-homogeneous or anisotropic material properties much more accurately than standard analytical methods. Additionally, it is not hampered by the limitations and challenges present in experimental techniques like strain gauge testing of real or model structures or photo elasticity. Therefore, dental stress analysis is especially well adapted to the finite element method. On an orthodontic patient, many force systems may be applied. With the finite element method, it is possible to apply different force systems analytically at any place and in any direction.[5] Hence, this FEM study was designed to evaluate and compare the effect of varying orthodontic forces on different orthodontic materials and surrounding dentofacial structures.

Materials and Methodology: It is an in-silico experimental study made with CT scan of maxillary dentition with its dentoalveolar complex and 64 bit computer with Windows 10. The various softwares used were as follows:

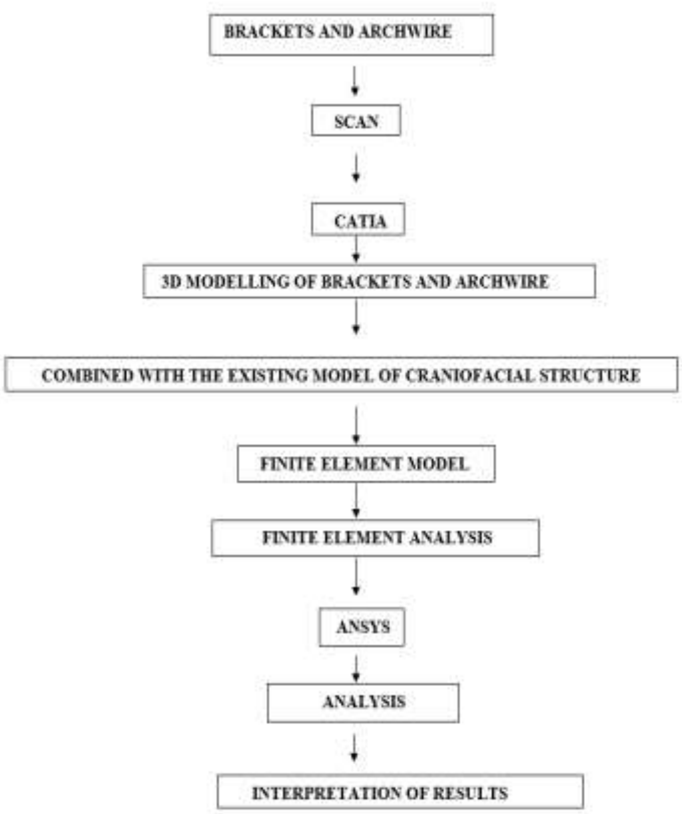
1. **DICOM:** Digital Imaging and Communications in Medicine is the international standard for medical images and related information (ISO 12052). It defines the formats for medical images that can be exchanged with the data and quality necessary for clinical use.
2. **MIMICS:** MIMICS (version 14.0, Materialise NV, Belgium) is a 3D Medical Image Processing and Editing Software, licensed and user-friendly application which translates scanner data into 3D CAD models, ensures effective, efficient work,

ISSN 2063-5346

analysis, visualization of radiological image data and simulation of surgical procedures. From the CT image the outlines and borders of teeth and bone required for constructing the Finite element model was obtained using the MIMICS software and a three-dimensional model was constructed by stacking the individual slices of CT images. This model was imported to CATIA V5 R19 for construction of surface and solid features of the geometric model.

3. **CATIA V5 R19:** The images obtained from MIMICS were in stereo lithographic format (STL format). The surface models of the tooth, periodontal ligament, cortical and cancellous bone were generated individually. These images were imported into CATIA V5 R19 software. These images were then refined and smoothened to produce accurate contours. The discontinuous surface was healed and joined together to form a continuous surface model. Then the surface model was converted into a solid model by volume meshing.
4. **ANSYS R17.2:** ANSYS offers a comprehensive software suite that spans the entire range of physics, providing access to virtually any field of engineering simulation that a design process requires. Three dimensional models of mini implant screws (2×10 mm, Stainless Steel, Bio-ray (Bio-Ray Biotech, Taipei, Taiwan) brackets and archwire (0.019×0.025), was constructed using CATIA software by scanning them. Then this model was combined with existing model of craniofacial structures including tooth, periodontal ligament and surrounding dentoalveolar structures. Force was applied from mini screw to hook. *Stress* occurring in the mini screw implant and the surrounding dentofacial structures was analysed using ANSYS software. The steps in this study were mentioned in figure 1.

Figure 1. Steps of FEM



ISSN 2063-5346

STEPS IN FINITE ELEMENT METHOD:

1. **Scanning:** Existing model of maxillary dentition with dentoalveolar complex from repository were selected for the study.
2. **Creation of Finite Element Model:** The DICOM Images obtained after scanning were converted to a finite element model using CATIA, MIMICS, ANSYS software.
3. **Finite Element Analysis:** First step involved is Pre processing, where existing CT Scan of the patient is converted into digital images with the help of MIMICS 14.0 software. Dimensions of mini-implant screws were obtained using scanned electron microscopy. Mini implants and archwires are constructed using CATIA V5 R19 software. Blue light scanner is used for scanning brackets. Maxillary dentition including, PDL, and surrounding Bone were modelled using MIMICS software's. Brackets, mini implant screws and archwires are modelled using CATIA software. Meshing of all the models is carried out using ANSYS software. ANSYS 15.0 software is used to carry out Finite Element Analysis. After applying Loads and Material Properties, solving is done. The Last and Final Step involves Post Processing where plots for *Stress* are obtained using ANSYS software. Three-dimensional model was constructed. This model was imported to Catia V5.

Figure 2. Geometrical model

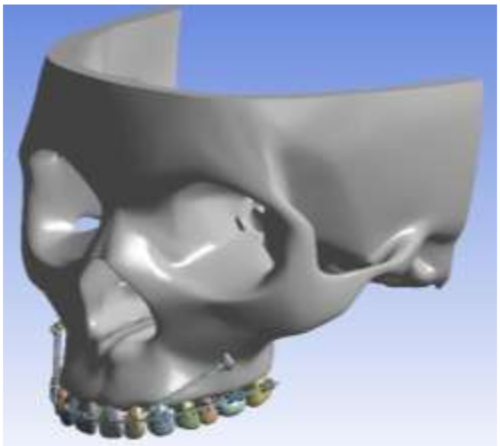
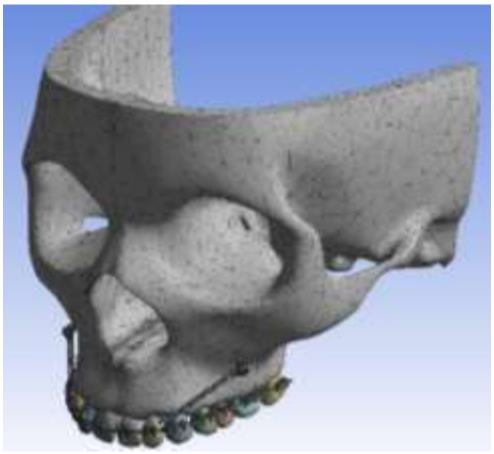


Figure 3. Mesh model



Results: Two mini implant screws, MBT Bracket System and Maxillary archwire were modelled using CATIA V5 R19 software. Pre-existing maxillary dentition along with surrounding craniofacial structures model was used. The models were meshed using ANSYS

ISSN 2063-5346

Software. Later all the nodes were imported to ANSYS software for analysis. Loads and material properties are applied for Finite Element Analysis. It is followed by step solving, for stress.

Stress produced in different orthodontic materials and surrounding craniofacial structures are recorded. Point of force application was from miniscrew to crimpable hook placed on the archwire between canine and lateral incisor. Stress generated for 150g of force were recorded. Graphs were plotted for Stress occurred on different orthodontic materials and surrounding dentofacial structures with different mode of retraction. The stress peek levels of various dentoalveolar structures were mentioned in table 1 and 2.

Table 1. Stress peek maximum force values

Load	Stress peek maximum
Stress Distribution in Complete System	0.13672
Stress Distribution in PDL	-4.47E-05
Stress Distribution in Tooth	4.56E-03
Stress Distribution in SKULL	2.05E-03
Stress Distribution in Implant	0.27361
Stress Distribution in Peri Implant Bone	0.11815

Table 2. Stress peek minimum force values

Load	Stress peek minimum
Stress Distribution in Complete System	-0.11083
Stress Distribution in PDL	-8.21E-04
Stress Distribution in Tooth	-7.15E-03
Stress Distribution in SKULL	-3.23E-03
Stress Distribution in Implant	-0.25801
Stress Distribution in Peri Implant Bone	-7.60E-02

Discussion: Orthodontic tooth movement (OTM)takes place by the coupling of bone resorption on the compressed side and by bone formation on the stretched side of the periodontal ligament (PDL) as a result of stress. Thus Prolonged treatment time leads to caries, periodontal problems. Root resorption and time burdensome for the patients[6]. In this aspect, to shorten the treatment time it is important to accelerate alveolar bone remodeling during orthodontic treatment. Low-energy laser irradiation and Magnetic fields[7] and injection of prostaglandin E2 (PGE2) and 1,25-(OH)2D3 during tooth movement, have been investigated[8].

Finite element analysis (FEA) is useful to determine the amount of stress, strain, and displacement in the dentoalveolar complex because of different loading conditions of force[9].

Section A-Research paper
ISSN 2063-5346

The FEM is a noninvasive, accurate method that permits detailed analyses of tooth movement and it provides with quantitative data that gives the physiologic reactions that occur after force application and reactions and interactions of individual tissue.

In this study, the FEA was done with MIMICS software which provided the outline and borders of teeth and bone structures. A three-dimensional model was constructed by stacking the individual slices of CT images. They were further refined with another software called CATIA V5 R19 which assist in construction of surface and solid features of geometric model. Three dimensional models of mini-implant screws (2×10 mm, Stainless Steel, Bio-ray (Bio-Ray Biotech, Taipei, Taiwan)), brackets, and archwire (0.019×0.025), were constructed using CATIA software by scanning them. Then this model was combined with existing model of craniofacial structures including tooth, periodontal ligament and surrounding dento-alveolar structures. Force of 150g was applied from mini screw to hook. The stress distribution occurring in the mini screw implant and the surrounding dentofacial structures were analysed using ANSYS software[10].

The present study was aimed at evaluating the effect of varying orthodontic forces on different orthodontic materials and surrounding dentofacial structures. The stress distribution in skull bone, pdl, tooth, implant and complete system showed similar values with no statistical significance. In this study, on comparing the stress distribution between the various dentofacial structures at maximum amplitude, the peek levels were maximum for tooth followed by skull and implant. PDL exhibited least stress distribution and the stress distribution of complete system at stress peek maximum was about 0.136N. In this study, on comparing the stress distribution between the various dentofacial structures at minimum amplitude, the stress peek levels were maximum for pdl followed by tooth and peri implant bone. Implant exhibited least stress distribution and the stress distribution of complete system at peek maximum was about 0.110N.

Conclusion: The present FEM study concluded that;

1) Application of varying orthodontic force produces maximum stress distribution in various dentofacial structures. While increasing the frequency of forces, there is significant decrease in stress distribution

3) The stress distribution in skull bone, pdl, tooth, implant and complete system showed similar values with no statistical significance.

References:

1. Tsichlaki, A.; Chin, S.Y.; Pandis, N.; Fleming, P.S. How long does treatment with fixed orthodontic appliances last? A systematic review. Am. J. Orthod. Dentofac. Orthop. 2016, 149, 308–318
2. Huang H, Williams RC, Kyrkanides S. Accelerated orthodontic tooth movement: molecular mechanisms. Am J Orthod Dentofacial Orthop. 2014;146:620–632.
3. Viecilli RF.Self-corrective T-loop design for differential space closure. Am J Orthod Dentofacial Orthop. 2006;129(1):48-53.
4. 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.
5. Yettram AL, Wright KW, Houston WJ. Centre of rotation of a maxillary central incisor under orthodontic loading. British journal of orthodontics. 1977 Jan;4(1):23-7.
6. Gainsforth BL, Higley LB. A study of orthodontic anchorage possibilities in basal bone. American Journal of Orthodontics and Oral Surgery. 1945 Aug 1;31(8):406-17.

ISSN 2063-5346

7. Tanne K, Sakuda M, Burstone CJ. Three-dimensional finite element analysis for stress in the periodontal tissue by orthodontic forces. Am J Orthod Dentofacial Orthop.1987 Dec;92(6):499-505.
8. Kusy RP, Whitley JQ. Friction between different wire-bracketconfigurations and materials. InSeminars in orthodontics 1997 Sep 1 (Vol. 3, No. 3, pp. 166-177). WB Saunders
9. Geramy A. Initial stress produced in the periodontal membrane by orthodontic loads in the presence of varying loss of alveolar bone: a three- dimensional finite element analysis. The European Journal of Orthodontics. 2002 Feb 1;24(1):21-33.
10. Gelgör İE, Büyükyılmaz T, Karaman AI, Dolanmaz D, Kalaycı A. Intraosseous screw–supported upper molar distalization. Angle Orthod.2004 Dec;74(6):838-50.