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 – 4, Issue – 3, May - 2021, Page No. : 229 - 235

Finite Element Analysis of Biomechanics of Total Arch Distalization of Maxillary Arch in Adult Skeletal Class II Pattern with Infrazygomatic Crest Bone Screws

[1]Dr. Sheetal Potnis, Professor, Dept. of Orthodontics and Dentofacial Orthopaedics, Sinhgad Dental College and Hospital, Pune

[2]Dr. N. G. Toshniwal, H.O.D, Dept. of Orthodontics and Dentofacial Orthopaedics, Rural Dental College, Loni

[3]Dr. Shilpa Pharande, Reader, Dept. of Orthodontics and Dentofacial Orthopaedics, Sinhgad Dental College and Hospital, Pune

[4]Dr. Rutuja Devadkar, P.G. Student, Dept. of Orthodontics and Dentofacial Orthopaedics, Sinhgad Dental College and Hospital, Pune

Corresponding Author: Dr. Sheetal Potnis, Professor, Dept. of Orthodontics and Dentofacial Orthopaedics, Sinhgad Dental College and Hospital, Pune

Citation of this Article: Dr. Sheetal Potnis, Dr. N. G. Toshniwal, Dr. Shilpa Pharande, Dr. Rutuja Devadkar, "Finite Element Analysis of Biomechanics of Total Arch Distalization of Maxillary Arch in Adult Skeletal Class II Pattern with Infrazygomatic Crest Bone Screws", IJDSIR- May - 2021, Vol. – 4, Issue - 3, P. No. 229 – 235.

Type of Publication: Original Research Article

Conflicts of Interest: Nil

Abstract

Aim: The aim of the study was to analyze the biomechanics of enmasse distalization of the maxillary arch in adult skeletal Class II pattern.

Materials and Methods: Finite Element Model analysis was done. Computer configuration used for the study 1) Hardware: A computer (DELL XPS System) with Intel i7 with 8-core processor, 8 GB RAM, 2 GB Graphics, loaded with Windows 7 operating system was used. 2) Software: FEA Pre-processing stage- Altair HyperMesh, FEA Solver- Altair Optistruct, FEA Post-processor- Altair Hyper View. Pretreatment CT Scan of adult patient with skeletal Class II pattern was obtained to generate CAD Model for this study. Geomagic Software was used to generated full Maxilla CAD Model. The FE Model was submitted to Altair Opti Struct Software for Finite Element Calculation.

Results and discussions: Post-Processing was done in Altair Hyper View Software. Von Mises stress results for teeth were calculated maximum and minimum stress results for teeth were also calculated.

Conclusion: By using a 3-dimensional finite element simulation the mechanics of total arch distalization of the maxillary arch were clarified.

Keywords: Class II pattern, Distalization, Finite element analysis, infrazygomatic crest bone screws, maxillary arch

Introduction

The aim of the study was to analyze the biomechanics of Enmasse distalization of the maxillary arch in adult skeletal Class II pattern.

The FEM is an engineering resource used to calculate stress and deformations in complex structures, and it has been widely applied in biomedical research. Finite element analysis was introduced into Orthodontics by Yettram et. al. in 1972.

Elements- 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.

Node- The elements that connect all characteristic points called node that lie on their circumference.

The objectives of this study were as follows:

- To analyze the force vectors acting on the maxillary arch during total arch distalization using infrazygomatic crest bone screws.
- To study the movement of the dentition in response to the forces in total maxillary retraction.

Materials

Finite Element Model was used.

Computer configuration used for the study:

1) Hardware: A computer (DELL XPS System) with Intel i7 with 8-core processor, 8 GB RAM, 2 GB Graphics, loaded with Windows 7 operating system was used.

2) Stage and software:

a) FEA Pre-processing- Altair Hypermesh

b) FEA Solver- Altair Optistruct

c) FEA post processor- Altair HyperView

3) Infrazygomatic bone screws 2 x 14 mm stainless steel (Favanchor)

4) MBT bracket prescription 0.022”x0.028” slot

5) E-chains and ligature ties

6) 19x25 stainless steel wire with hooks.

Method

Pre-processor-Finite element model is built in this step. This includes conversion of CAD to FE model. Applying Boundary conditions.

Solver- Calculations for Numerical Simulation

Post processor- Read and interpret Results

- Pre-Processing: As the name says this is the process before we run FEA calculations. At this step the model was constructed. Mesh was generated. Brackets, loops along with arch wires were placed at predefined locations. Finally. Boundary conditions were applied.

Software used at Pre-processing stage was Altair Hyper Mesh.

- FEA Solver: This is the software which processes the input and calculates results for the Finite Element Model.

Software used as Solver stage was Altair Opti Struct.

- Post-Processing**:** This is the stage where Results are seen, extracted, reviewed and stored. Results like, Reaction force, Displacements etc. are all processed and extracted at this step. This data is then stored in excel sheets or word files for further interoperations.

Software used for post-processing of results was Altair Hyper View.

Essential Steps in Fem

- Select the type of analysis
- Discretization
- Develop element matrices and equating
- Derivation of overall equations/ matrices
- Imposition of boundary status
- Application of loads
- Post processing of results

Generation of CAD Model

CT scan was obtained to generate CAD Model for this study. Geomagic Software was used to generate full Maxilla CAD Model

Fig 1a: Generation of CAD Model

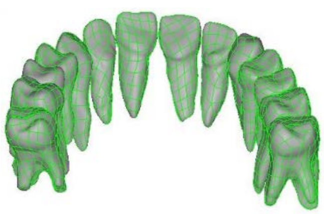
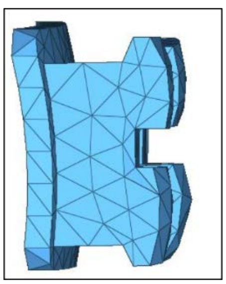


Fig 1b: Generation of CAD Model

Fig 2: Brackets Construction of MBT prescription brackets and tubes



FEM model of MBT prescription bracket (0.022”x0.028”)

Fig 3: Archwires

A standard coordinate system was constructed-

X-axis -bucco-palatal direction

Y-axis -antero-posterior direction

Z-axis - superior-inferior direction

Fig 4: Teeth added

Fig 5: PDL Developed

Fig 6: Jaw Bone

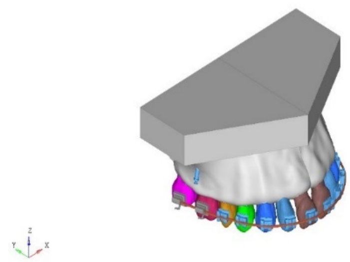


Fig 7: Full Model

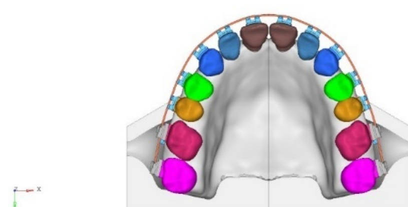


Fig 8: IZC screw added

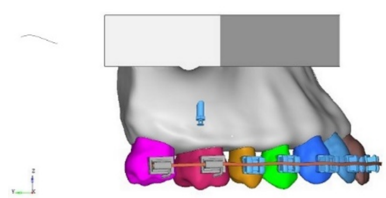


Fig 9a: Finite element model

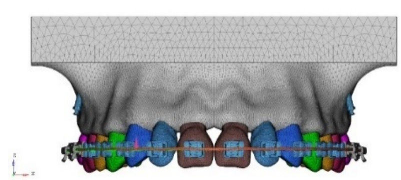


Fig 9b: Finite element model

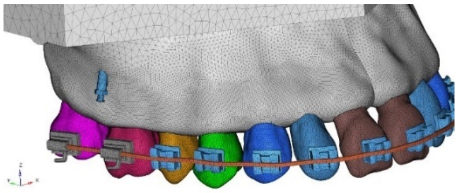


Fig 10: FEM on Class II for maxillary arch distalization using infrazygomatic crest bone screws

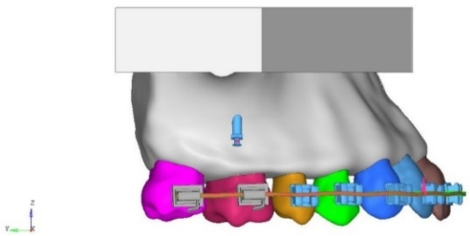


Table 1: Meshing Details

Component	No. of Nodes	No. of elements
Tooth (CMI)	305956	1609838
Periodontal Ligament (PDL)	80973	128881
Alveolar bone	177620	983126
Enamel	99702	158694
Cementum	80107	127322
Defect 1Bd	3687	15113
Defect 2Bd	4538	19261
Defect 3Bd	7956	34371

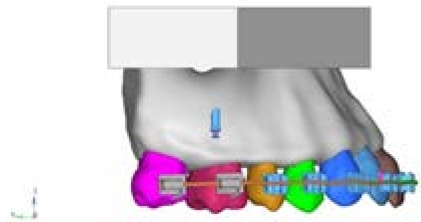
Node and Element Count

Meshing Strategy

Subdivide i.e discretize the complex geometry into suitable set of smaller "elements" of finite dimensions (2D or 3D).

- The points connecting two or more discrete elements are called as nodes or nodal points. The corner nodes are called primary external nodes.
- The additional nodes which occur on the sides of the elements are called secondary external nodes. The secondary nodes have fewer displacement than corner nodes.
- Collection of nodes and element result in formation of element mesh that could be in shape of tetrahedran prism and hexahedron.
- Number, size and type of element are decided. Practical knowledge and judgement are needed to limit the number of elements to minimum amount conductive to acceptable results.

Fig 11: Boundary conditions



Calculations

The FE Model was submitted to Altair OptiStruct Software for Finite Element Calculation

Develop element matrices and equating

- Specify material properties to the elements and obtain algebraic equation defining stiffness for each element.
- Stiffness matrix (K) will relate the forces acting on the structure and displacement resulting from these forces in following manner.

  Fig 12: Equation

$$[K]\{u\} = \{F\} \Longrightarrow \{u\} = [k]^{-1}\{F\}$$

K = Stiffness matrix; **u** = Deflection; **F** = Force

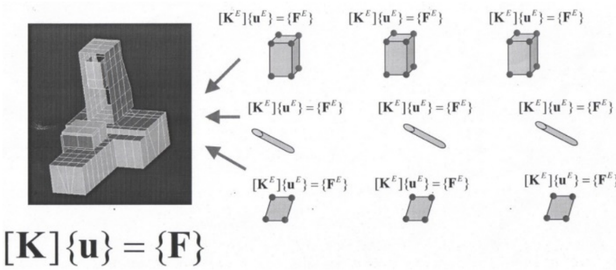
Derivation of overall equation and matrices

Displacement at a node has to be same for all adjacent elements.

Combine element matrices to obtain one master equation called Global stiffness matrix

Thus FEM uses the concept of piecewise polynomial interpolation by connecting elements together, the field quantity becomes interpolated over the entire structure in piecewise fashion.

Fig 13: Global stiffness matrix



Results and Discussions

- The way Results are Presented in this report is explained in this section
- Post-Processing was done in Altair HyperView Software

Fig 14a: Von Mises stress results for teeth

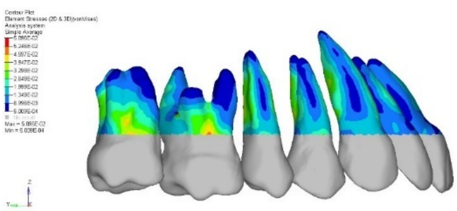


Fig 14b: Von Mises stress results for teeth

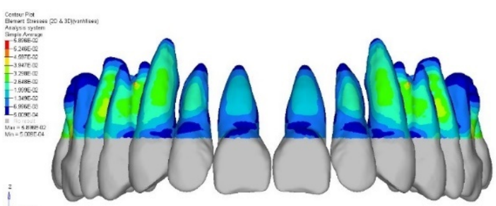
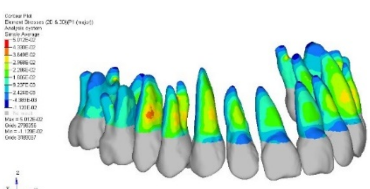


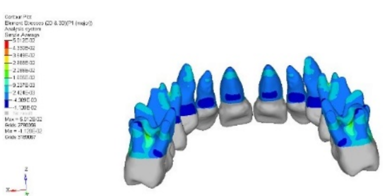
Fig 15a: Maximum principal stress results for teeth



P1 Stress = Max. Principal Stress shows tension stress across the root of the teeth

Front end of the roots are in +ve stress

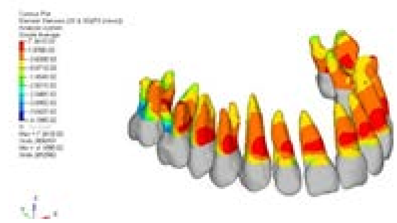
Fig 15b: Maximum principal stress results for teeth



P1 Stress = Max. Principal Stress shows tension stress across the root of the teeth

Rear end of the root are in –ve stress (Blue Colour)

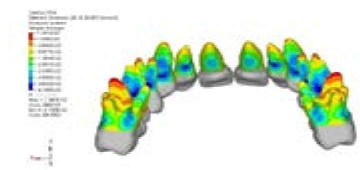
Fig 16a: Minimum principal stress results for teeth



P3 Stress = Min. Principal Stress shows compression stress across the root of the teeth

Front end of the roots are in +ve stress

Fig 16b: Minimum principal stress results for teeth



P3 Stress = Min. Principal Stress shows compression stress across the root of the teeth

Rear end of the root are in –ve stress (Blue to yellow colour)

Fig 17 :Stress results for jaw bone

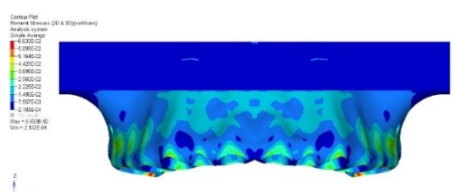


Fig 18a Signed von. mises stress

Counter plot example

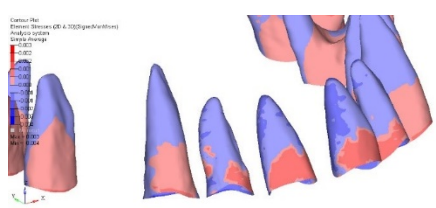
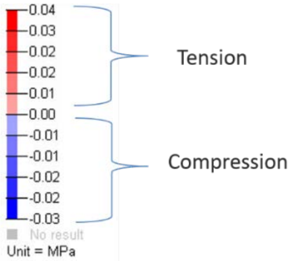
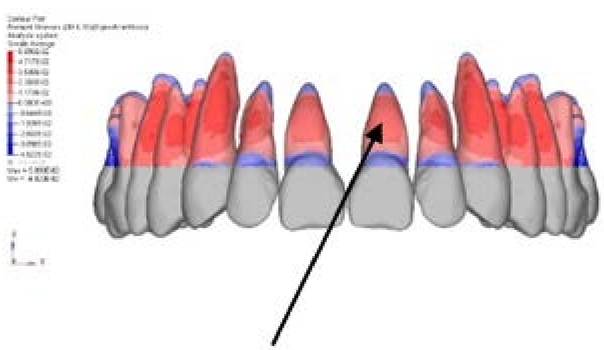


Fig 18b: Signed von. Mises stress



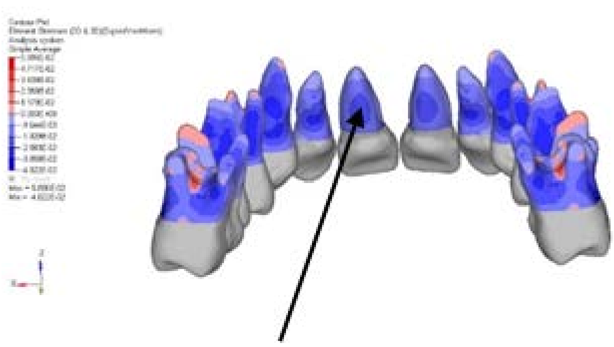
- Simple Von. Mises Stress might not always give clear idea when it comes to compression & tension stress. Signed Von. Mises is used in such cases
- As name suggests; Signed Von. Mises stress has sign convention. Positive (ve+) for Tension Stress and Negative (ve-) for Compressive Stress.
- Contour plot has 2 colour tone; Red for Tension & Blue for Compression
- All red areas in the contour plot is tension & blue is compression

Fig 19:Tension stress indicate resorption



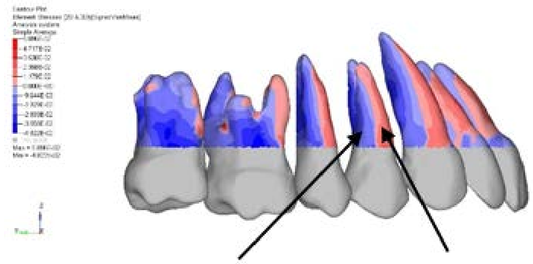
Red plot at front indicate tension side

Fig 20: Compression stress indicate absorbption



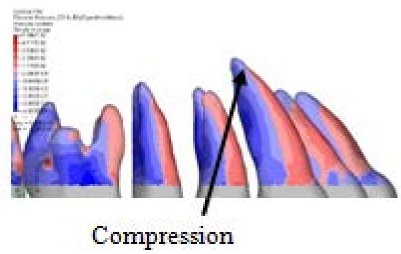
Blue plot at front indicate absorbption

Fig 21: Stress results for teeth



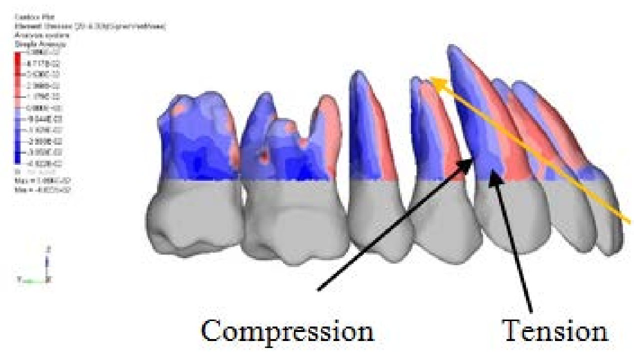
- As seen in above image rear side of all tooth roots are in compression stress where front end is tension side which means at front we got resorbption & at rear of tooth we got absorption of bone
- This indicates that tooth moment is in posterior direction

Fig 22: Stress results for teeth



- Tips of all root are in compression. Suggesting intrusion + distillization

Fig 23: Stress results for teeth



Compression & Tension stress across the teeth helps conclude that overall tooth moment is as shown by yellow arrow

Conclusion

By using a 3-dimensional finite element simulation the mechanics of total distalization of the maxillary dentition were clarified. The following conclusions were reached:

1. Distalization of the entire arch could be achieved
2. Rotation of the arch in the clockwise direction.
3. Intrusion of posterior teeth
4. Extrusion of anterior teeth
5. Contraction of arch in the posterior region

References

1. Cornelis, M.A. and De Clerck, H.J.: Biomechanics of skeletal anchorage, Part 1: Class II extraction treatment, J. Clin. Orthod. 40:261-269, 200
2. De Clerck, H.J. and Cornelis, M.A.: Biomechanics of skeletal anchorage, Part 2: Class II nonextraction treatment, J. Clin. Orthod. 40:290-298, 2006.
3. De Clerck, H.J., Timmerman H. M. and Cornelis, M.A: Biomechanics of skeletal anchorage, Part 3: Intrusion, J. Clin. Orthod. 42(5):270-278, 2008.
4. Jung MH, Kim TW. Biomechanical considerations in treatment with miniscrew anchorage. Part 1: the sagittal plane. Journal of Clinical Orthodontics: JCO. 2008 Feb;42(2):79.
5. Jung MH, Kim TW. Biomechanical considerations in treatment with miniscrew anchorage. Part 2: the horizontal and transverse planes. Journal of clinical orthodontics: JCO. 2008 Mar;42(3):144-8
6. Jung MH, Kim TW. Biomechanical considerations in treatment with miniscrew anchorage. Part 3: clinical cases. Journal of clinical orthodontics: JCO. 2008 Jun;42(6):329-7
7. Park H-S. Implant orthodontics: an interactive approach to sleletal anchorage. In: Vinod Krishnan ZeD, editor. Integrated Clinical Orthodontics. Wiley-Blackwell; 2012.
8. Papadopoulos MA. Orthodontic treatment of Class II malocclusion with miniscrew implants. Am J Orthod Dentofacial Orthop 2008; 134:604.e1-16: [discussion 604-5].
9. Zhang N, Bai Y, Li S. Treatment of a Class II Division 1 malocclusion with miniscrew anchorage. Am J Orthod Dentofacial Orthop 2012;141:e85-93
10. Liu TC, Chang CH, Wong TY, Liu JK. Finite element analysis of miniscrew implants used for orthodontic anchorage. Am J Orthod Dentofacial Orthop. 2012 Apr;141(4):468-76.
11. Kawamura, J., Tamaya, N. A finite element analysis of the effects of archwire size on orthodontic tooth movement in extraction space closure with miniscrew sliding mechanics. Prog Orthod. 20, 3 (2019).
12. Ghorbanyjavadpour F, Kazemi P, Moradinezhad M, Rakhshan V. Distribution and amount of stresses caused by insertion or removal of orthodontic miniscrews into the maxillary bone: A finite element analysis. International Orthodontics. 2019 Dec;17(4):758-768.
13. Alrbata, Raed & Momani, Moath & Tarawneh, Ahmad & Ihyasat, Ayman. (2015). Optimal force magnitude loaded to orthodontic microimplants: A finite element analysis. The Angle orthodontist. 86. 10.2319/031115-153.1