



Journal Homepage: -www.journalijar.com
**INTERNATIONAL JOURNAL OF
 ADVANCED RESEARCH (IJAR)**

Article DOI:10.21474/IJAR01/6431
 DOI URL: <http://dx.doi.org/10.21474/IJAR01/6431>



RESEARCH ARTICLE

EVALUATION OF STRESS AND DEFORMATION ON PROSTHETIC IMPLANTS DURING MINI-PLATE ASSISTED CANINE AND EN-MASSÉ RETRACTION IN LINGUAL ORTHODONTICS: A FINITE ELEMENT *in silico* STUDY.

Dr. Shamu Rapheal¹, Dr. Rohan Mascarenhas², Dr. Satish Shenoy³ and Dr. Akhter Husain⁴.

1. Post graduate, department of orthodontics and dentofacial orthopedics, Yenepoya Dental College, Mangalore, Karnataka.
2. Professor, department of orthodontics and dentofacial orthopedics, Yenepoya Dental College, Mangalore, Karnataka.
3. Professor and head, department of aeronautical engineering, Manipal Institute of Technology, Manipal, Karnataka.
4. Professor and head, department of orthodontics and dentofacial orthopedics, Yenepoya Dental College, Mangalore, Karnataka.

Manuscript Info

Manuscript History

Received: 03 December 2017
 Final Accepted: 05 January 2018
 Published: February 2018

Keywords:-

Implants, Finite element method,
 Lingual orthodontics, Mini-plates.

Abstract

Lingual appliances marked a great leap forward in aesthetic orthodontics. Due to their distinct biomechanics compared to conventional orthodontics, special care must be taken in their application. As mini-plate and implants are new in orthodontics, the in depth knowledge about their mechanical changes are limited. In this study, stress and deformation on prosthetic implant was evaluated during mini-plate assisted canine and en-masse retractions in lingual orthodontics using finite element method. The results obtained indicated that the stress and deformation values were higher during mini-plate assisted canine retraction than en-masse retraction.

Copy Right, IJAR, 2018,. All rights reserved.

Introduction:-

INVISIBILITY of the appliance is one of the major benefits of lingual system; and is an ideal option for adult patients who are concerned about aesthetics. In adults, the probability of prosthetic replacements could be higher due to loss of natural teeth. Prosthetic implants are well accepted substitute to natural teeth. Mini-plates are one of the best devices used for absolute anchorage in adult orthodontics. The desire to have complete control over anchorage is universal among orthodontists. The force systems that are used on an orthodontic patient can be complicated. However, the finite element method makes it possible to analytically apply various force systems at any point and in any direction. FEM has been used successfully to study the application of forces to single and multi-tooth systems, for the changes during various tooth movements using fixed mechanotherapy[1]. This study evaluates deformation and stress related changes in prosthetic implants of maxillary arch during mini-plate assisted canine and en-masse retractions in lingual orthodontics. Evaluation is done by loading 1N, 2N and 3N forces on a FEM model of maxilla during canine and en-masse retractions assisted by mini-plate.

Corresponding Author:-Shamu Rapheal.

Address:-Post graduate, department of orthodontics and dentofacial orthopedics, Yenepoya Dental College, Mangalore, Karnataka.

Methodology:-

This study is conducted to determine the stress and deformational changes on implants during mini-plate assisted canine and en masse retraction in lingual orthodontics. The materials required for the study included; bit computer with windows 7, CT scan of upper anterior dentition with its dentoalveolar complex and various softwares (CHART I)

Chart I:-Softwares Used In The Study

MIMICS (version 14.0, Materialise NV, Belgium) (Materialize Interactive Medical Image Control System)
CATIA V5 R19 (for surface and solid modelling)
ANSYS R17.2 – General Post processing software(Structural)

Computerized Tomography scan of upper anterior dentition with dentoalveolar complex of patient were selected for the study after obtaining Institutional ethical clearance. The DICOM (Digital Imaging Communications in Medicine) Images obtained after scanning were converted to a finite element model using CATIA, MIMICS & ANSYS software. Pre Processing, where CT Scan of the patient is converted into digital images with the help of MIMICS 14.0 software. Dimensions of implant, mini-plate and miniscrews were obtained using Scanned Electron Microscopy (Fig.1). Implants, mini-plates, mini-screws and arch wires are constructed using CATIA V5 R19 software. Blue light scanner is used for scanning brackets. Upper anterior dentition including, PDL, and surrounding Bone are modeled using MIMICS software's. Brackets, implants, mini-plates and arch wires are modeled using CATIA software. Meshing of all the models is carried out using ANSYS software.

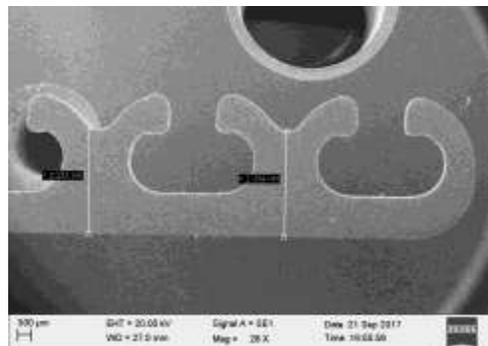


Fig. 1:-Scanned electron microscopy images of mini-plate

ANSYS R17.2 software is used to carry out Finite Element Analysis. After applying Loads and Material Properties, solving is done for Stress, Strain and Deformation.

The Last and Final Step involves Post Processing where plots for Stress Strain and Deformation are obtained using ANSYS software.

Finite element method basically can be divided into 7 steps, which include; Construction of the geometric model, Conversion of geometric model to a Finite Element model, Material property data representation, Defining the boundary condition, Loading Configuration, Solving the system of linear algebraic equation and Interpretation of the results

In this study, previously created model of upper anterior dentition with its dentoalveolar complex of an adult male after obtaining the ethical clearance from Yenepoya university ethical committee (YUEC 315/01/2016)

CT images for the upper anterior dentition were obtained in DICOM (Digital Imaging Communications in Medicine) format with Philips CT scanner in the regime of 140KV, 800mA, 512 by 512 matrix, field of view (FOV) 14cm by 14cm and slice thickness of 0.7mm. CT images are only in a series of two dimensional images, and it cannot be directly used for finite element modeling. The images thus obtained were imported in MIMICS software for processing. 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 R19 for construction of surface and solid features of

the geometric model. The images thus obtained from MIMICS were in stereo lithographic format (STL format). The surface models of the teeth, periodontal ligament, cortical and cancellous bone were generated individually. Brackets, Mini-plate, Implants, Mini-screws and arch wire were modeled using CATIA software. All these images were imported into CATIA V5 R19 software. These images were then refined and smoothed 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.

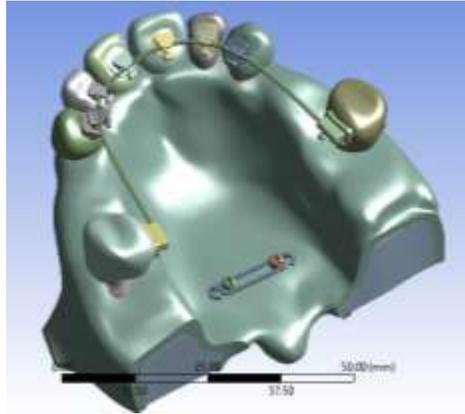


Fig. 2:-Maxillary arch with anterior teeth incorporating lingual brackets and archwire, implants, crowns and mini-plate

The finite element modeling is the representation of geometry in terms of a finite number of elements and nodes. The finalized model (Fig. 2) was then imported into the Finite Element Analysis software ANSYS workbench version 14.5 in .iges format. The created volumes of geometric model were individually divided into finite number of elements; these elements were connected to adjacent elements with the help of nodes. The element shape in the model was a solid tetrahedral with 3 degree of freedom each unstrained nodes. The finite element model (Fig. 3) consisted of 237918 nodes and 127861 elements, and three degree of freedom (TABLE I, II, III & IV). The models were meshed with higher order tetrahedral elements of element size 0.5mm. This measurement of mesh was chosen after a trial of multiple mesh sizes since it produced the best and most accurate patterns. The continuous nature of the meshes was checked by manual inspection.

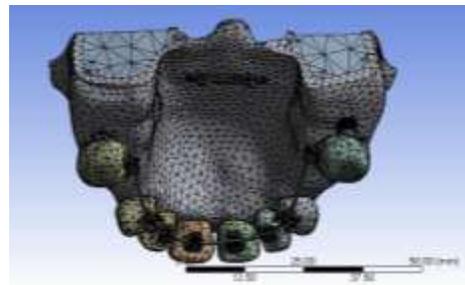


Fig. 3:-Meshed model of maxillary arch with anterior teeth incorporating lingual brackets & arch wire, implants, crowns and mini-plate

Table I:-Number Of Elements And Nodes Used For Dentition

Teeth		Nodes	Elements
Central Incisor	Right	1574	833
	Left	1819	989
Lateral Incisor	Right	1769	930
	Left	2025	1073
Canine	Right	2579	1407
	Left	1629	866

Table ii:-Number Of Elements And Nodes Used For Brackets And Tubes

Brackets and tubes		Nodes	Elements
Central Incisor Bracket	Right	7545	4199
	Left	7604	4261
Lateral Incisor Bracket	Right	7048	3961
	Left	6013	3359
Canine Bracket	Right	7545	4199
	Left	7604	4261
Molar tubes	Right	2912	1557
	Left	3178	1713

Table 3:- Number Of Elements And Nodes Used For Pdl

PDL		Nodes	Elements
Central Incisor	Right	4898	2460
	Left	5038	2520
Lateral Incisor	Right	4702	2384
	Left	4393	2206
Canine	Right	6614	3293
	Left	4555	2278

Table 4:-Number Of Elements And Nodes Used For Bone

Bone	Nodes	elements
Cortical Bone	53527	28572
Spongy Bone	33063	18681

Table 5:-Number Of Elements And Nodes Used For Implants, Abutments, Crowns, Mini-Screws, Mini-Implant, Implant Screw And Mini-Plate

Implant, mini-plate and wire		Nodes	Elements
Implant	Right	9217	4883
	Left	9418	4999
Abutment	Right	3389	1783
	Left	3466	1840
Crown	Right	4550	2689
	Left	4845	2898
Mini-screw	Right	2912	1557
	Left	3890	1890
Implant- screw	Right	1762	835
	Left	1790	857
Mini-plate	Palatal	5315	2965
Wire	Palatal	1216	428

Total nodes: 224820

Total elements: 121117

Quality Criteria Used:

Warpage > 5

Aspect ratio >10

Skew > 45

Jacobian > 0.7

Tet collapse > 0.25 (for tetras)

Tria Min angle > 20

Max angle > 120 QUAD Min angle > 45

Max angle > 135

The area of concern is prosthetic implant. This was modeled with good quality high number of elements to capture the result gradients accurately. The number of elements decides the quality of the results and the system used for solving. It is very difficult to have ideal shaped elements embedded in the complex non-specific geometry. Hence the entire Finite Element mesh has to meet a set of quality criteria which decides the nature of the outcome. The quality criteria included was:

Element twisting:-

Shape collapse:-

The result of the Finite Element Analysis is directly proportional to the mesh quality. If the mesh quality is good, the convergence problems are minimal. The number of elements is directly proportional to computational time. Hence careful judgments were made while meshing the significant parts.

Material properties were assigned for different structures such as bone, teeth and periodontal ligament (TABLE VII). The material properties used in this study was taken from experimental data from previous studies.

Table 6:-Material Properties Used In The Study

Material	Young's modulus(n/mm^2)	Poisson's ratio
Tooth	20300	0.30
PDL	0.667	0.45
Cortical bone	13700	0.26
Cancellous bone	790	0.30
Stainless Steel	860	0.30–0.31
Titanium(Grade V)	880	0.31–0.37

In this study all the tissues are assumed to be isotropic, i.e., the properties of the material are same in all directions.

The model was restrained at the superior border of the maxilla in order to avoid any motion against the loads imposed on the dentoalveolar structures. The final model was confirmatory from an engineering point of view for this study. It is very important to keep in mind that the finite element model will give the results based upon the nature of modeling system and for that reason, the procedure for modeling is most important.

Using MIMICS software maxillary anterior dentition was modeled. Using CATIA V5 R19 software archwires, CAD/CAM bracket systems, mini-plates and implants are modeled. Then all the geometric models were imported to ANSYS software for analysis. In this study, the aim was to determine the stress related changes generated on prosthetic implants during mini-plate assisted canine and en-masse retraction in lingual orthodontics.

The sequential application of the above steps leads to a system of simultaneous algebraic equations where the nodal displacements are the unknowns.

As a last step the results obtained after processing was displayed and graphical output were made. This is a pictorial representation of the results, which gets displayed in a color-coded manner. The maximum and minimum stress and deformation in the implants were recorded. Interpretations of the various analysis were carried out at this stage. The colors represent the magnitude of stress/deformation at that area. The palate of colors along with values is depicted on the side of the images. This enables a quick interpretation of the results. The colours vary from red to blue. Positive values indicate tensile stress and negative values indicates compressive stress. Unit of Stress is MPa. (One MPa is equal to one N/mm^2).

Results:-

Using CATIA V5 R19 software implants, mini-plates with two mini-screws, 7th Generation Lingual Bracket System (Anterior teeth) and Maxillary archwire were modeled. Maxillary arch with anterior dentition along with PDL were modelled using MIMICS software. The models were meshed using ANSYS software. Later all the modes were imported to ANSYS software for analysis. Loads and material properties are applied after Finite Element Analysis. It is followed by step solving, where we solve for stress and deformation.

Stress and deformation produced in implants are recorded during mini-plate assisted canine and en-masse retraction

at various predetermined force levels (Fig. 5-17). During mini-plate assisted canine retraction and en-masse retraction, point of force application was from hook of the mini-plate to hook of canine bracket and on the arch wire between canine and lateral incisor respectively (Fig. 4). Stress and deformation values generated for 1N, 2N and 3N forces were recorded (TABLE VII & VIII). Graphs were plotted for Force Vs Stress and Force Vs Deformation for comparing canine retraction and en-masse retraction (Fig 17 and Fig 18).

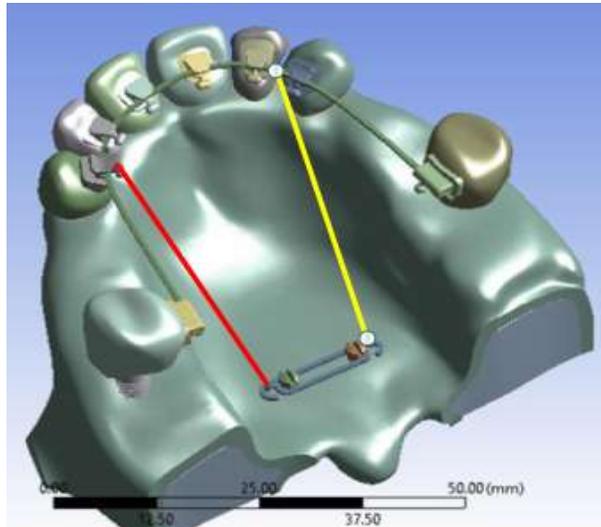


Fig. 4:-Image showing line of force during Canine retraction (Red) and en-masse retraction (Yellow)

The graphical comparisons conveyed that for every increase in the force value there is a linear progression of stress and deformation during canine and en-masse retraction. The observations made from the graphical representations showed stress on implant is higher during canine retraction than en-masse retraction. Deformation of implant is higher during mini-plate assisted canine retraction than mini-plate assisted en-masse retraction. The stress and deformation pattern generated were represented by different colours from blue to red which are expressed MPa (N/mm²) and mm respectively.

Table Vii:- Stress Values Of Implant For 1N, 2N And 3N Forces During Mini-Plate Assisted Retractions

Force value	Stress (Unit: MPa)	
	En-masse Retraction	Canine Retraction
1N	0.55816	0.59628
2N	1.1163	1.1926
3N	1.6859	1.7999

Table Viii:- Deformation Values Of Implant For 1N, 2N And 3N Forces During Mini-Plate Assisted Retractions

Force value	Deformation (unit: mm)	
	En-masse Retractions	Canine Retraction
1N	0.000084386	0.000090529
2N	0.00016877	0.00018106
3N	0.00025491	0.00027333

Stress on implant during canine retraction

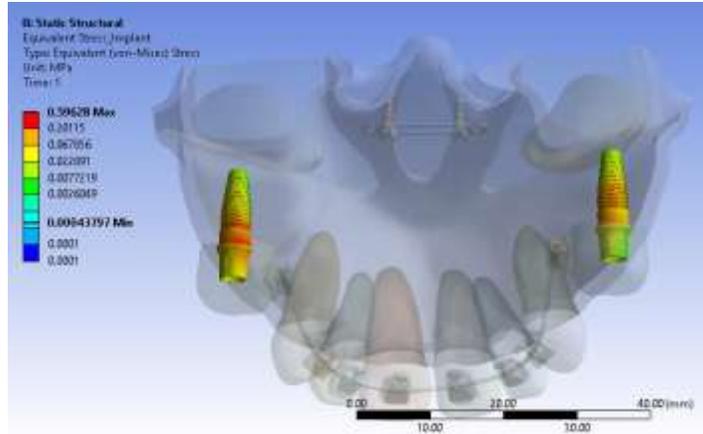


Fig. 5:-For 1N force, maximum stress of 0.59628 MPa was observed at the neck of the left side implant and minimum stress of 0.00043797 MPa was observed at the apical region of the implant

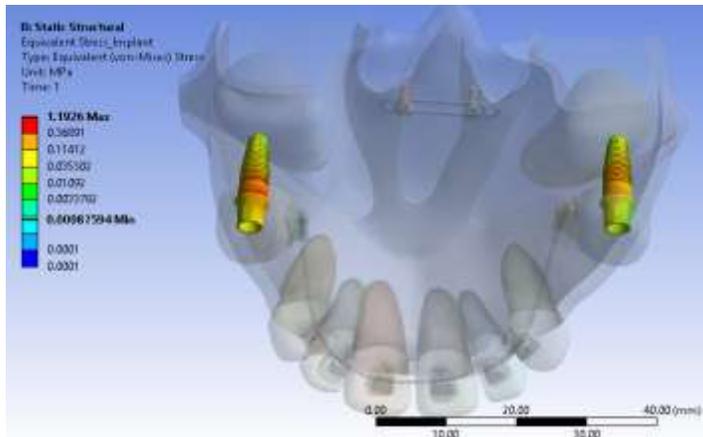


Fig. 6:-For 2N force, maximum stress of 1.1926 MPa was observed at the neck of the left side implant and minimum stress of 0.00087594 MPa was observed at the apical region of the implant

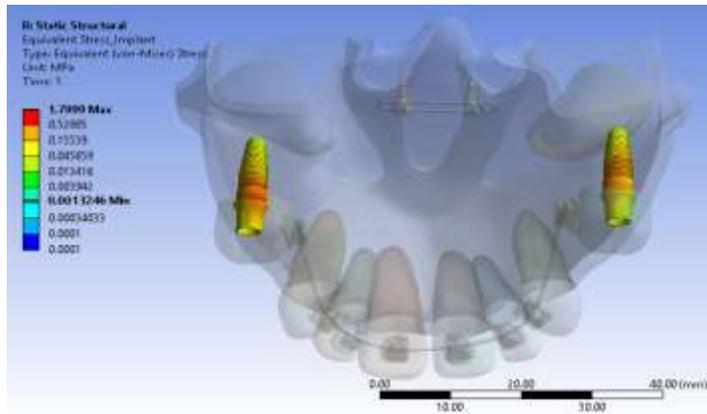


Fig. 7:-For 3N force, maximum stress of 1.7999 MPa was observed at the neck of the left side implant and minimum stress of 0.00013246 MPa was observed at the apical region of the implant

Stress on implant during en-masse retraction

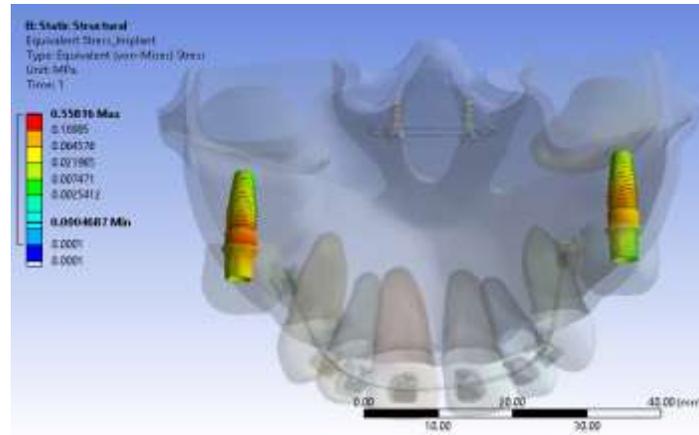


Fig. 8:-For 1N force, maximum stress of 0.55816 MPa was observed at the neck of the left side implant and minimum stress of 0.0004687 MPa was observed at the apical region of the implant

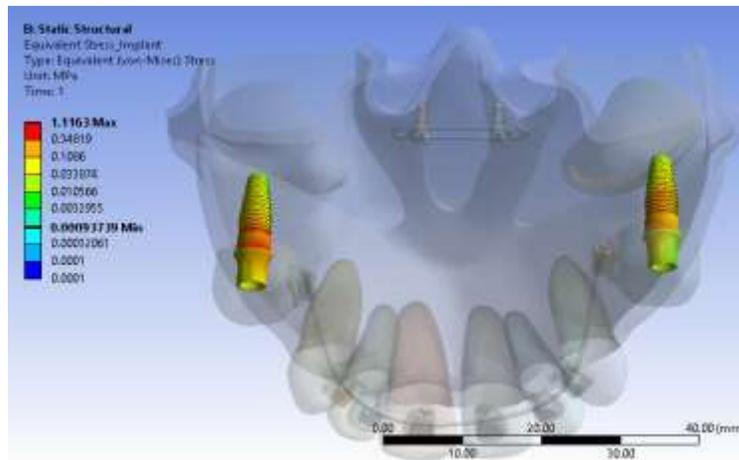


Fig. 9:-For 2N force, maximum stress of 1.1163 MPa was observed at the neck of the left side implant and minimum stress of 0.00093739 MPa was observed at the apical region of the implant

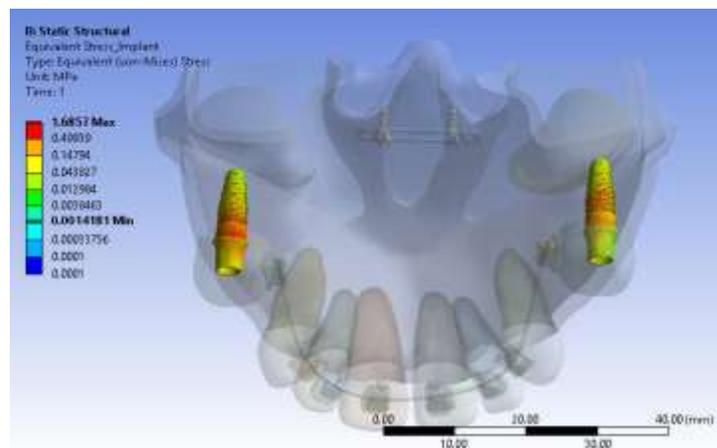


Fig. 10:-For 3N force, maximum stress of 1.6857MPa was observed at the neck of the left side implant and minimum stress of 0.00014181MPa was observed at the apical region of the implant

Deformation of implant during canine retraction

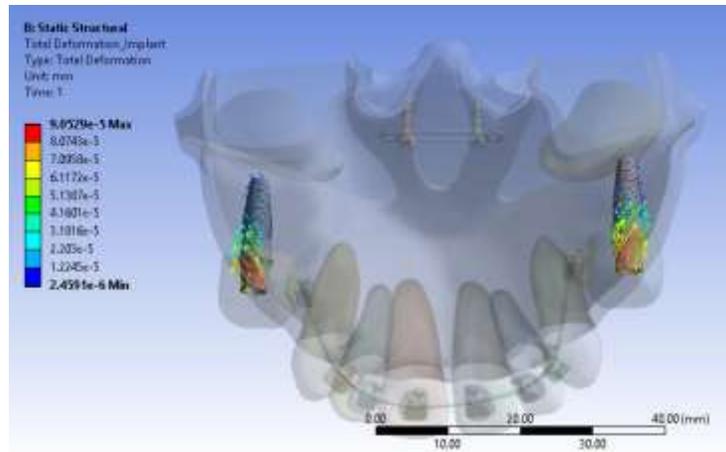


Fig. 11:-For 1N of force, Maximum deformation of 0.000090529 mm was observed at the abutment region and minimum deformation of 0.0000024591 mm was observed at the apical region

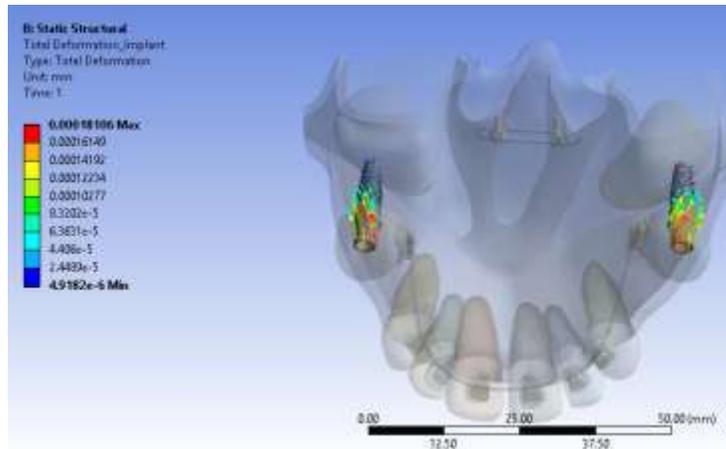


Fig. 12:-For 2N of force, Maximum deformation of 0.00018106 mm was observed at the abutment region and minimum deformation of 0.0000049182mm was observed at the apical region

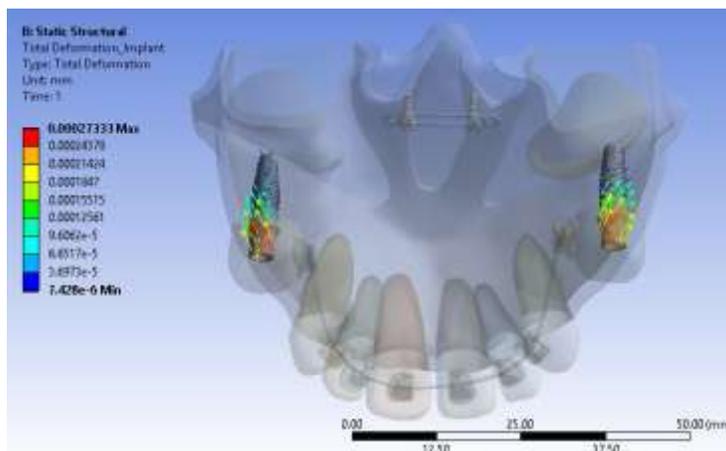


Fig. 13:-For 3N of force, Maximum deformation of 0.00027333 mm was observed at the abutment region and minimum deformation of 0.000007428 mm was observed at the apical region

Deformation of implant during en-masse retraction

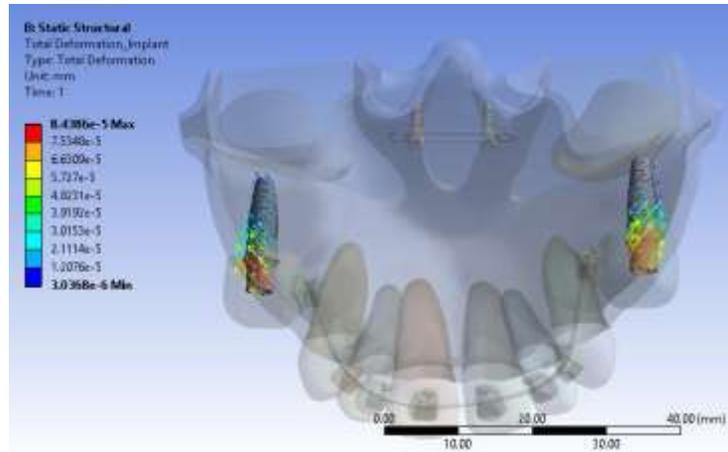


Fig. 14:-For 1N of force, Maximum deformation of 0.000084386 mm was observed at the abutment region and minimum deformation of 0.0000030368mm was observed at the apical region of the implant

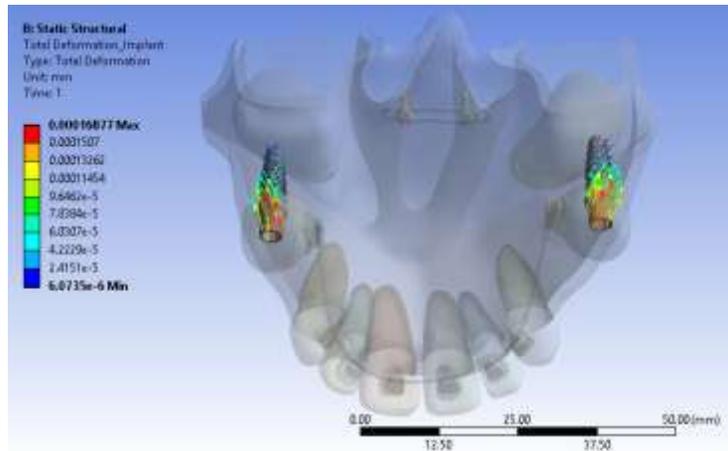


Fig. 15:-For 2N of force, Maximum deformation of 0.00016877 mm was observed at the abutment region and minimum deformation of 0.0000060735mm was observed at the apical region of the implant

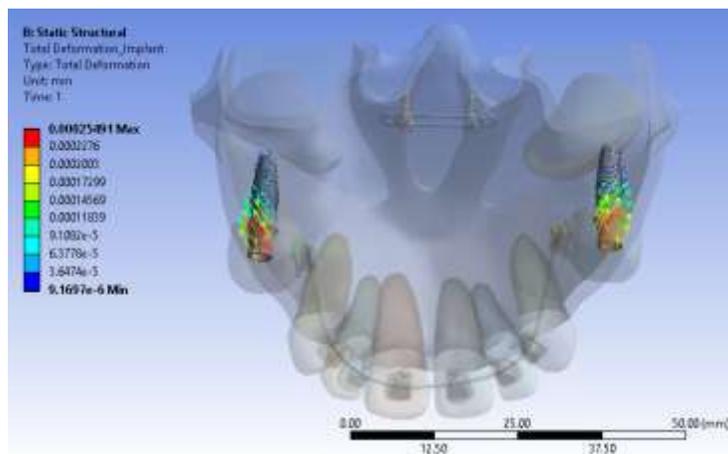


Fig. 16:-For 3N of force, Maximum deformation of 0.00025491 mm was observed at the abutment region and minimum deformation of 0.0000091697mm was observed at the apical region of the implant

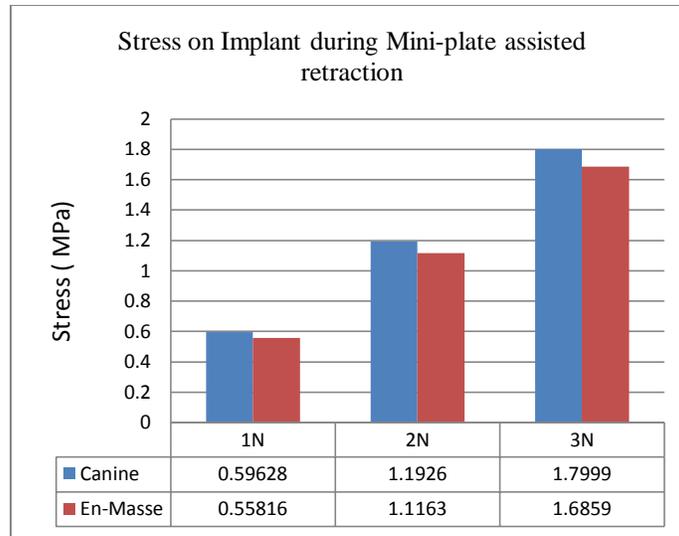


Fig. 17:-Comparison of stress on implants during mini-plate assisted canine and en-masse retractions for various force values

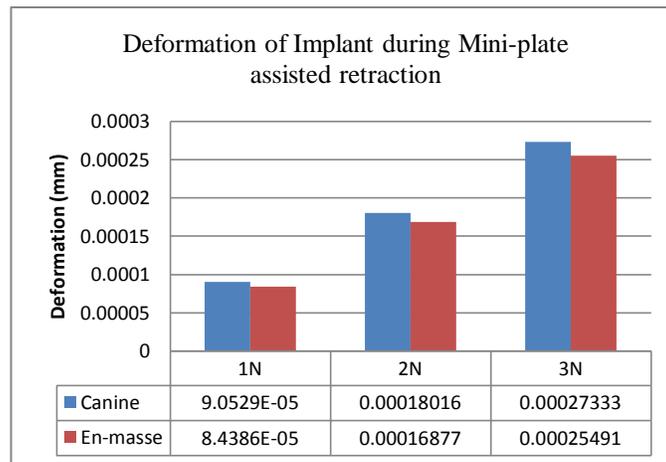


Fig. 18:- Comparison of deformation of implants during mini-plate assisted canine and en-masse retractions for various force values

Discussion:-

One of the most important determinants in successful orthodontic treatment is anchorage control. Orthodontic anchorage is categorized into maximum, moderate and minimum anchorage, depending on the amount of movement of the anchoring unit during force application [2]. Of the three types of anchorage, maximum anchorage is usually most desirable and, at the same time, the most difficult to achieve. Absolute anchorage is the term used to describe the anchoring unit that remains unmoved under orthodontic forces. With the advent of mini-plate and other temporary anchorage devices the possibility of absolute anchorage is realized[3].

Orthodontic treatment requires optimum force to bring about maximum tooth movement with minimal damage to the root, periodontal ligament, and alveolar bone [4]. This will also bring changes in the adjacent prosthesis if present. These changes are generally given less importance as the stand away from the mainstream of treatment. Hence, the quantification and direction of force is very important during orthodontic treatment. Therefore, the stressed state of the implant needs to be studied in order to understand the effect of different forces.

In this study, stress and deformations of implants during mini-plate assisted canine and en-masse retractions were evaluated. For every increase in the force value, a linear increase in the stress and deformation values were observed. Observations of stress and deformation changes at various sites were tabulated (TABLE VII & VIII).

Comparing canine and en-masse retractions stress and deformation values were lower during mini-plate assisted en-masse retraction than canine retractions.

Conclusion:-

This study is conducted using three dimensional finite element analysis to evaluate the stress related changes in implants during mini-plate assisted canine and en-masse retractions for forces of 1N, 2N and 3N. The results showed that there is stress and deformation are generated on implants during retraction and they are higher during mini-plate assisted canine retractions than en-masse retractions. Mini-plates and implants being a new armamentarium in orthodontics, the in depth knowledge about their mechanical changes are limited. Finite Element Method is a very efficient tool to analyse and understand their mechanical changes.

References:-

1. Yettram AL, Wright KWJ, Houston WJB. Center of rotation of a maxillary central incisor under orthodontic loading. *Br J Orthod* 1977; 4:23-27.
2. Nanda ravindra- Biomechanics and esthetic strategies in clinical orthodontics .Elsevier health sciences.
3. Rungcharassaeng K, Kan JY, Caruso JM. Implants as absolute anchorage. *J Calif Dent Assoc.* 2005 Nov; 33(11):881-8.
4. Mascarenhas R. A comparative study of forces in labial and lingual orthodontics using finite element method. 2015; 49(1):15-18.