ORIGINAL RESEARCH PAPER
Volume - 13 | Issue - 03 | March - 2024 | PRINT ISSN No. 2277 - 8179 | DOI : 10.36106/ijsr
INTERNATIONAL JOURNAL OF SCIENTIFIC RESEARCH

THE EFFECTS OF STRESS DISTRIBUTION BY TWO DIFFERENT ORTHODONTIC MINISCREW IMPLANTS WITH DIFFERENT ANGULATIONS: A 3 DIMENSIONAL FINITE ELEMENT ANALYSIS.

Dentistry

Dr Pooja Ashok Magar PG student.

Dr. Amit Ajmera Professor & AMP; PG Guide.

KEYWORDS

INTRODUCTION:

In recent years Ortho Mini-Implants (OMIs) have become popular because of their versatility, minimal surgical invasiveness, low cost and increase in the overall treatment spectrum.[1,2] The optimal insertion angle has an important role in anchorage, biomechanical consideration, risk of damage to adjacent teeth and also delivers enhanced surface contact area between the bone and OMIs. Based on the aim of tooth movement, OMIs may be used in various directions, thus understanding how the force is transmitted to the OMIs and cortical and trabecular bone would be valuable to clinicians. The present study aims to measure the distribution of stress patterns in the OMIs and in the bone.

This study aims to investigate the roles of bone quality, angulation on the biomechanics of orthodontic miniscrew implants(DENTOS & BIORAY) by using finite element analysis.

In this study, we incorporated a finite element approach and factorial analysis to determine the biomechanical effects of the exposure length of the mini-implant, the insertion angle, and the direction of orthodontic forces. This was the original research conducted on FEM models and didn't require any approval from ethical committee,hence no approval was taken from the ethical committee.

To reduce the failure rate of OMIs (10-30%), several biological and biomechanical factors associated with their performance have been studied.[3-5] It can be speculated that the inclination angle in which it is inserted, is a factor of major relevance.[4-6]

Previous studies using FEA have shown the quality of trabecular bone in the region of OMI insertion is not critical for its stability, however, a layer of cortical bone at least 1 mm thick is crucial.[7] On the other hand, the thinner the cortical bone, the larger the tension the OMIs will cause to the bone, increasing the risk of implant failure.[8-9] It has been shown that even the thick cortical bone layer of the palate region is incapable of supporting forces greater than 480 gf.[10] The shape and geometry (macro structure) of an OMI are directly connected with the amount of tension it will induce into the bone.[11-12] The region in which the highest tensions to the cortical bone are detected is the first thread of the OMI.[13-14] Furthermore, the angle at which the OMI is inserted might alter this parameter.

Materials:-

- DENTOS Cylindrical TAD 1.5mm x 8mm and Bio_Ray Cylindrical TAD 1.5mm x 8mm was procured for 3D laser scanning
- 3D Digital replica of the TAD was created using Rexcan III 3D White light Scanner
- Computer Aided Design (CAD) was constructed using the information of scanned Models.
- A Finite Element Model (FEM) will be created with the help of a CAD model
- Bone model was generated directly in FEA software (Altair Hypermesh).

Method-

Following are general steps involved in the finite element model preparation are as follows
1. Construction of the geometric model of the mandible bone around the OMIs & complete the assembly.
2. Incorporation of material properties of tooth structure and periodontium.
3. Loading configuration.

Construction of the geometric model of OMIs & Bone.

In this study, the geometry of 3D finite element model of DENTOS Cylindrical & Bio_Ray Cylindrical OMI was generated through physical laser scanning. This data were exported to 3D image processing and editing software- 3D-Doctor (Able Software Corp., Lexington, Massachusetts, USA), stereolithographic (STL) Bone model was made in Altair HyperMesh (Troy, Michigan, United States). Complete model was assembled in Altair HyperMesh software.

Incorporation Of Material Properties Of Tooth Structure And Periodontium

The material properties of teeth, cortical bone, cancellous bone, OMIs were the average values reported in literature. A detailed table with all material data is listed below. All materials employed for the finite element model study were taken to be isotropic, homogenous, and linearly elastic.

Loading Configuration

Validation Of Model:

This was a stage for trial RUN of FEA. It was a stage for element quality check. Element qualities like Warp age, Aspect Ratio, tet Collapse were checked & required modification were done in the model to improve the element quality.

Local re-meshing in some areas was considered to improve the overall mesh quality of the Finite Element Model. All this was done to ensure generated mesh/element quality is within acceptable limit for FEA Solver.

Force Application:

Load was applied as follows:
· 100 N.mm Torque

Generation Of Cad Model

Two implants (TADs) were considered for this study. As follows:
DENTOS Cylindrical TAD 1.5mm x 8mm

BioRay Cylindrical TAD 1.5mm x 8mm

CAD Model for TADs was obtained from a 3D laser scanning device. Actual models of TAD were sent for 3D scanning and a 3-dimensional CAD model was generated. CAD Model images & specifications of the TAD are as mentioned below

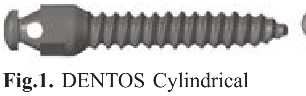


Fig.1. DENTOS Cylindrical TAD 1.5mm x 8mm

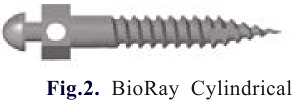
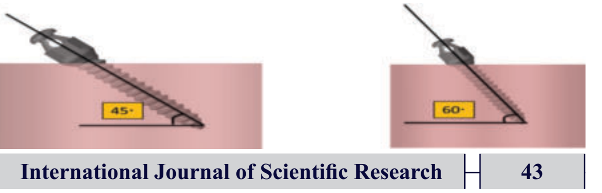


Fig.2. BioRay Cylindrical TAD 1.5mm x 8mm

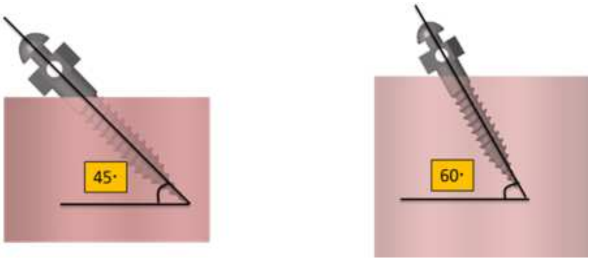
Placement of TAD Model-

Screw was inserted in the Bone Block at 2 different angulations as shown below (45 &60). Centreline sections for 2 models are placed side by side for comparison.

Following Are First Two Models For DENTOS Cylindrical TAD 1.5mm x 8mm



Following Are First Two Models For BioRay Cylindrical TAD 1.5mm x 8mm



RESULT-

Bone Stress Distribution

Generally, the amount of stress for force direction in all angulations (45°and 60°) was greater in the cortical bone than the trabecular bone. The greatest amount of stress in cortical bone was at the entrance of the miniscrew to the bone; the stress levels reduced gradually toward the tip of the miniscrew.

Miniscrew Stress Distribution

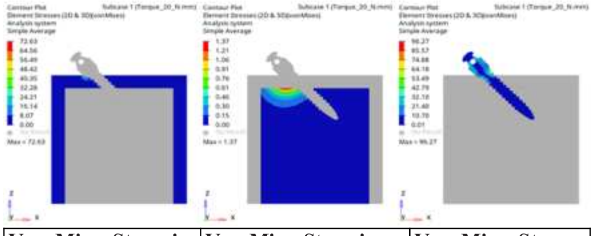
The peak von Mises stress in the miniscrew was at the smallest diameter of the shank, especially in the part inserted in the soft tissue. Generally, toward the tip of miniscrew, this value was decreased. For the part of the mini-screw located within the cortical bone, the greatest amount of stress was in the narrowest part of the shank.

Table:1

Models	TAD Details	TAD Insertion Angle (°)	Load
1	DENTOS Cylindrical TAD	45	100 N.mm Torque
2	DENTOS Cylindrical TAD	60	100 N.mm Torque
3	Bio_Ray Cylindrical TAD	45	100 N.mm Torque
4	Bio_Ray Cylindrical TAD	60	100 N.mm Torque

Model 1-Temporary Anchorage Device DENTOS at 45 degree under the load of 100N.mm distribute more stress on bone.

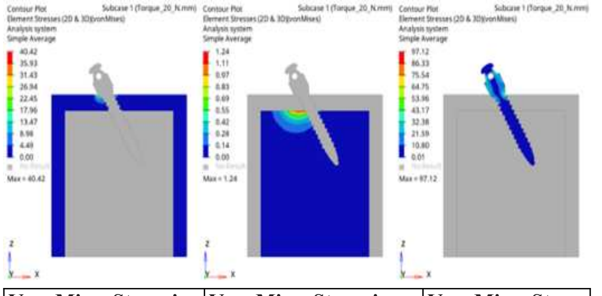
Model	TAD Details	TAD Insertion Angle (°)	Load
1	DENTOS Cylindrical TAD	45	100 N.mm Torque



Von. Mises Stress in Cortical Bone (MPa)	Von. Mises Stress in Medullary Bone (MPa)	Von. Mises Stress in TAD (MPa)
72.63	1.37	96.27

Model 2-Temporary Anchorage Device DENTOS at 60 degree under the load of 100N.mm distribute less stress on bone as compare to 45 degree angulation

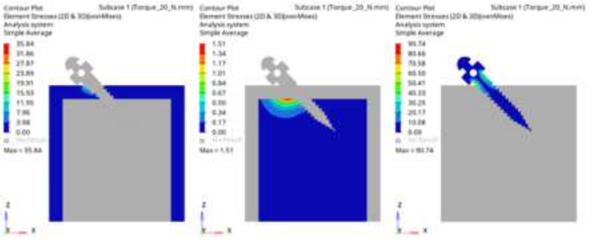
Model	TAD Details	TAD Insertion Angle (°)	Load
2	DENTOS Cylindrical TAD	60	100 N.mm Torque



Von. Mises Stress in Cortical Bone (MPa)	Von. Mises Stress in Medullary Bone (MPa)	Von. Mises Stress in TAD (MPa)
40.42 MPa	1.24 MPa	97.12 MPa

Model 3-Temporary Anchorage Device Bioray at 45 degree under the load of 100N.mm distribute less stress on bone.

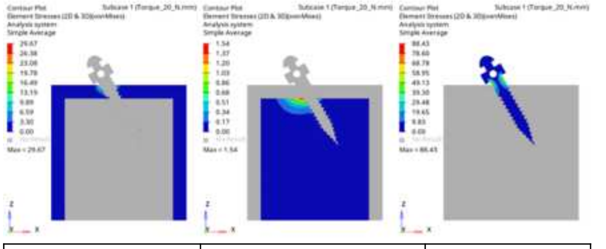
Model	TAD Details	TAD Insertion Angle (°)	Load
3	Bio_Ray Cylindrical TAD	45	100 N.mm Torque



Von. Mises Stress in Cortical Bone (MPa)	Von. Mises Stress in Medullary Bone (MPa)	Von. Mises Stress in TAD (MPa)
35.84 MPa	1.51 MPa	90.74 MPa

Model 4-Temporary Anchorage Device Bioray at 60 degree under the load of 100N.mm distribute less stress on bone.

Model	TAD Details	TAD Insertion Angle (°)	Load
4	Bio_Ray Cylindrical TAD	60	100 N.mm Torque



Von. Mises Stress in Cortical Bone (MPa)	Von. Mises Stress in Medullary Bone (MPa)	Von. Mises Stress in TAD (MPa)
29.67 MPa	1.54 MPa	88.43 MPa

DISCUSSION-

In recent years Ortho Mini-Implants (OMIs) have become popular because of their versatility, minimal surgical invasiveness, low cost and increase in the overall treatment spectrum.[1,2] The optimal insertion angle has an important role in anchorage, biomechanical consideration, risk of damage to adjacent teeth and also delivers enhanced surface contact area between the bone and OMIs. Finite element model analysis (FEMA) provides an effective computational tool, which offers a stable and non-invasive alternative method for measuring the response of the structures to the applied external loads under certain boundary conditions. This study was conducted to evaluate two different types of implants(Dentos and Bioray) for their effects of force directions on the stress distribution in different parts engaged in the process of loading.[15] Applying the 100-Nmm force with 45° and 60° angles showed that the greatest amount of stress was in the cortical bone which was significantly higher than that of the trabecular bone.[16]

Excessive bone stress might cause local bone resorption. Other reasons for the difference between the amounts of von Mises stress in various studies are different mesh design and size, as well as the shape and diameter of the screws, different Young's modulus, and deliberation of friction. In this study, like most other studies, the maximum von Mises stress levels with 100-Nmm horizontal force were less than the yield strength of cortical bone.[17]

In previous study, Liu et al.found that a horizontal load would induce much more stress than a vertical load. So logically vertical stress provides less stress than pure horizontal force.

The highest amount of stress was seen in the area with the smallest diameter just above of the entrance of miniscrew to the cortical bone.[18] Therefore, the risk of failure is higher in this area. This can be justified with the second moment of inertia of a cylinder that shows the peak stress is inversely proportional to the third power of the diameter. These findings were different from the result achieved by Singh et al. [9] due to the fact that they did not consider a space for the soft tissue and reported the maximum stress to be located in the neck of miniscrew.

According the results of this study, it can be suggested that the part of the miniscrew which is inside the soft tissue should have a larger diameter relative to the portion that is located inside the cortical bone. Simultaneously, the thread width of the soft tissue area should be reduced compared to those in contact with the bone.[19]

The thread shapes did not have a significant impact on the miniscrew deflection under the horizontal loading. This value decreased with the reduction of the angle of force because the amount of horizontal force was reduced as previously described.

Like other finite element studies, this study had some limitations in the simulation. The structures in the models were assumed to be linear, homogeneous, and isotropic; while real bone is neither homogeneous nor isotropic, but we used these assumptions for simplicity and to compensate for the lack of information on the bone behavior. The geometry of the bone block was simplified to a rectangular block instead of a jaw section. The soft tissue was not simulated; although its thickness was deliberated. We assumed that the friction coefficient between the miniscrew and bone was 0.2; it might be different for cortical and trabecular bones. Since the thread shape might affect the insertion of miniscrew in the bone, further studies on this subject are recommended.

CONCLUSION-

In this FEM study the force applied by OMI on bone with different angulation is measured. At angle of 45 degree and 60 degree. It was found that 60 degree angulation distribute less stress than 45 degrees. Comparison between DENTOS and BIORAY implant indicated that the BIORAY mini screw implant at 60 degree gives better result as far as stability and bone stress distribution is concerned.

REFERENCES-

1] Mauro Cozzani et al(2020)The ideal insertion angle after immediate loading in Jeil, Storm, and Thunder miniscrews: A 3D-FEM study.Cozzani M, Nucci L, Lupini D, Dolatshahizand H, Fazeli D, Barzkar E, Naeini E, Jamilian A. The ideal insertion angle after immediate loading in Jeil, Storm, and Thunder miniscrews: A 3D-FEM study. Int Orthod. 2020 Sep;18(3):503-508. doi: 10.1016/j.ortho.2020.03.003. Epub 2020 May 6. PMID: 32387220.

2] Crismani AG, Bertl MH, Celar AG, Bantleon HP, Burstone CJ. Miniscrews in orthodontic treatment: review and analysis of published clinical trials. Am J Orthod Dentofacial Orthop. 2010 Jan;137(1):108-13. http://dx.doi.org/10.1016/j.ajodo.2008.01.027. PMid:20122438

3] 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. http://dx.doi.org/10.1016/j.ajodo.2011.11.012. PMid:22464529

4] Lin TS, Tsai FD, Chen CY, Lin LW. Factorial analysis of variables affecting bone stress adjacent to the orthodontic anchorage mini-implant with finite element analysis. Am J Orthod Dentofacial Orthop. 2013 Feb;143(2):182-9. http://dx.doi.org/10.1016/j.ajodo.2012.09.012. PMid:23374924

5] Miyawaki S, Koyama I, Inoue M, Mishima K, Sugahara T, Takano-Yamamoto T. Factors associated with the stability of titanium screws placed in the posterior region for orthodontic anchorage. Am J Orthod Dentofacial Orthop. 2003 Oct;124(4):373-8. http://dx.doi.org/10.1016/S0889-5406(03)00565-1. PMid:14560266

6] Cheng SJ, Tseng IY, Lee JJ, Kok SH. A prospective study of the risk factors associated with failure of mini-implants used for orthodontic anchorage. Int J Oral Maxillofac Implants. 2004 Jan-Feb;19(1):100-6. PMid:14982362

7] Yu W, Park HS, Kyung HM, Kwon OW. Dynamic simulation of the self-tapping insertion process of orthodontic microimplants into cortical bone with a 3-dimensional finite element method. Am J Orthod Dentofacial Orthop. 2012 Dec;142(6):834-41

8] Motoyoshi M, Inaba M, Ono A, Ueno S, Shimizu N. The effect of cortical bone thickness on the stability of orthodontic mini-implants and on the stress distribution in surrounding bone. Int J Oral Maxillofac Surg. 2009 Jan;38(1):13-8. http://dx.doi.org/10.1016/j.ijom.2008.09.006. PMid:18963818.

9] Holberg C, Winterhalder P, Rudzki-Janson I, Wichelhaus A. Finite element analysis of mono- and bicortical mini-implant stability. Eur J Orthod. 2014 Oct;36(5):550-6. http://dx.doi.org/10.1093/ejo/cjt023. PMid:23598610

10] Lombardo L, Gracco A, Zampini F, Stefanoni F, Mollica F. Optimal palatal configuration for miniscrew applications. Angle Orthod. 2010 Jan;80(1):145-52. http://dx.doi.org/10.2319/122908-662.1. PMid:19852654.

11] Singh S, Mogra S, Shetty VS, Shetty S, Philip P. Three-dimensional finite element analysis of strength, stability, and stress distribution in orthodontic anchorage: a conical, self-drilling miniscrew implant system. Am J Orthod Dentofacial Orthop. 2012 Mar;141(3):327-36. http://dx.doi.org/10.1016/j.ajodo.2011.07.022. PMid:22381493

12] Luiz NE, Jacomini A Fo, Lima LAC, Ciuccio RL, Soares MAD, Coutinho LL. Optimization design of orthodontic screws aiming at increasing the mechanical strength. Innov. Implant. J., Biomater. Esthet. 2010 Aug;5(2): 30-34.

13] Marcé-Nogué J, Walter A, Gil L, Puigdollers A. Finite element comparison of 10 orthodontic microscrews with different cortical bone parameters. Int J Oral Maxillofac Implants. 2013 July-Aug;28(4):e177-89. http://dx.doi.org/10.11607/jomi.2447. PMid:23869375.

14] Motoyoshi M, Inaba M, Ueno S, Shimizu N. Mechanical anisotropy of orthodontic mini-implants. Int J Oral Maxillofac Surg. 2009 Sept;38(9):972-7.

15] Suzuki A, Masuda T, Takahashi I, Deguchi T, Suzuki O, Takano-Yamamoto T. Changes in stress distribution of orthodontic miniscrews and surrounding bone evalu- ated by 3-dimensional finite element analysis. Am J Or- thod Dentofacial Orthop. 2011; 140: e273-e280.

16] Huang HL, Chang CH, Hsu JT, Fallgatter AM, Ko CC. Comparison of implant body designs and threaded designs of dental implants: a 3-dimensional finite element analysis. Int J Oral Maxillofac Implants. 2007 Jul-Aug;22(4):551-62. PMID: 17929515

17] Misch CE. Dental implant prosthetics. 3rd ed. St Louis: Mosby; 2005. p.322–347.

18] Ivanoff CJ, Gröndahl K, Sennerby L, Bergström C, Lekholm U. Influence of variations in implant diame- ters: a 3- to 5-year retrospective clinical report. Int J Oral Maxillofac Implants. 1999; 14: 173-180.

19] Miyamoto I, Tsuboi Y, Wada E, Suwa H, Iizuka T. Influence of cortical bone thickness and implant length on implant stability at the time of surgery--clinical, pro- spective, biomechanical, and imaging study. Bone. 2005; 37: 776-780.