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 – 5, Issue – 2, April - 2022, Page No. : 395 - 402

Evaluation of different force effects on bone surrounding the mini implant - A fem study

[1]Dr. Naveen Kumar Ramagoni, Professor, Department of Pedodontics and Preventive Dentistry, Navodaya Dental College and Hospital, Raichur 584103, Karnataka, India.

[2]Dr. K Raja Sigamani, Professor, Rajah Muthiah Dental College, Chidambaram, Tamilnadu.

[3]Dr. Bhaskar, Professor, Rajah Muthiah Dental College, Chidambaram, Tamilnadu.

[4]Dr. Kurinchi Kumaran, Professor, Rajah Muthiah Dental College, Chidambaram, Tamilnadu.

[5]Dr. Arafath Mohammed, Professsor, Rajah Muthiah Dental College, Chidambaram, Tamilnadu.

[6]Dr. S. N. Reddy Duvvuri, Associate Professor, Rajah Muthiah Dental College, Chidambaram, Tamilnadu.

Corresponding Author: Dr. Naveen Kumar Ramagoni, Professor, Department of Pedodontics and Preventive Dentistry, Navodaya Dental College and Hospital, Raichur 584103, Karnataka, India.

Citation of this Article: Dr. Naveen Kumar Ramagoni, Dr. K Raja Sigamani, Dr. Bhaskar, Dr. Kurinchi Kumaran, Dr. Arafath Mohammed, Dr. S. N. Reddy Duvvuri, “Evaluation of different force effects on bone surrounding the mini implant - A fem study”, IJDSIR- April - 2022, Vol. – 5, Issue - 2, P. No. 395 – 402.

Type of Publication: Original Research Article

Conflicts of Interest: Nil

Abstract

Objective: The objective of the study was to evaluate the effect of varying loads of forces at different angulation at various levels on bone using finite element method (FEM).

Methods: A 3-dimensional model of the bone was constructed to simulate force magnitudes and directions. Finite element study was conducted with horizontal forces of 100g, 150g and 200 g applied to the bone at angulations of 90° and 120°. von Mises stress, strain and deformation values were then evaluated using the ANSYS software.

Results: Both stress and deformation increased with increasing the amount of loading force. These 2 indexes were linearly proportional to the force magnitude and produced the highest values when the force was 200 g applied at 120° to the long axis of the miniimplant. The peak deformation and von Mises stress was concentrated on the bone at the cervical level under 200 g at 120°.

Conclusion: The von Mises stress values in the bone were found to be lowest under 100 g force at 90°. Higher initial loading and horizontal force should be avoided on the cervical level of mini implant for better stability. The direction and the amount of orthodontic force had a significant effect on cortical bone stress with highest on 200g force at 120°.

Keywords: mini-implant, stress, strain, deformation, finite element method.

Introduction

Major challenge for a dentist during orthodontic treatment is anchorage. Hence mini implants have been emerging in replacing traditional methods by skeletal anchorage. [1] These are placed in the jaw bone between the roots of a teeth to serve as anchorage for the forces being applied.[2] Mini implants are advantageous because of their miniature size, easier insertion and removal, low cost, and ability to withstand immediate or early loading post surgery. However, mini implant loosening seems to be the practical issue in the success of the procedure.[3]Stress generated on bone around the mini implants influence their success rate. Factors such as the length and diameter of an implant and its direction of insertion in bone can affect the stress generated on the bone surrounding the implant. [4]

Finite element analysis results give insight about stress distribution and biomechanical changes in the implant–bone system. This method gives favourable degree of reliability and accuracy without the risks and expenses of implantation.[5]

The purpose of this finite element study was to estimate the effect of varying loads of forces at various angulations on bone. This will help to determine an optimal orthodontic force that can be loaded safely on mini implant to achieve adequate primary stability, and thus reduce the failure of mini implant in orthodontics.

Methodology

A FEM was created using a software (ANSYS 13 version 14.5 Creo 3.0). The FEM was composed of these elements: 1- MI model (diameter, length, and screw); 2- modeling of cortical and cancellous bones; 3- FEM of bone when MI is placed into bone at 90° angulation; and 4- Young's modulus and Poisson's ratio for all constituent structures under experiment.

To verify influences of the varying loads of forces at various angulations on bone, 12 three-dimensional FEMs were conducted. Orthodontic MI made of pure titanium (diameter - 2.6 mm; length - 10 mm; thread ridge height - 0.33 mm; thread pitch - 0.8 mm) was modeled. For the ease of modelling and based on the classical theory of elasticity, it was assumed that the constituent material was isotropic and homogeneous. The behaviour of the constituent material of FEM was quantified by Poisson's ratio and Young's modulus. The material properties for MI and bone are given in Table 1.

Table 1: Material properties of constituent materials

Materials	Young's modulus (MPa)	Poisson's ratio
Titanium	110,000	0.35
Cortical bone	14,000	0.30
Cancellous bone	1,370	0.30

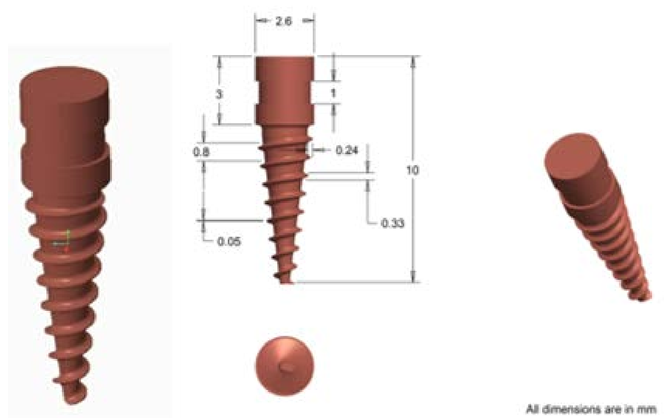

Fig 1: Schematic representation of implant design used in this finite element study.

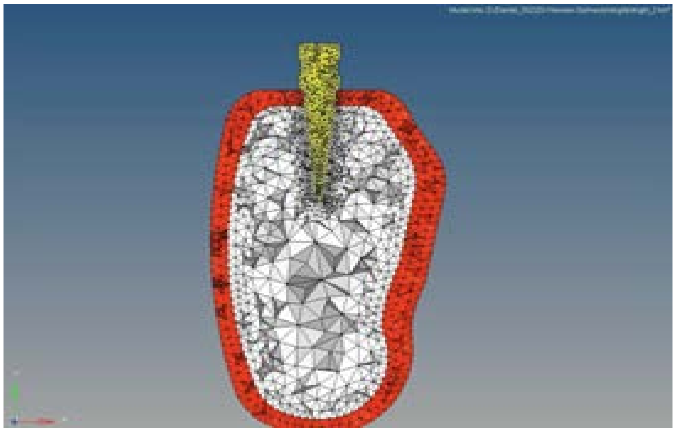

Fig 2: Final model with colour coding

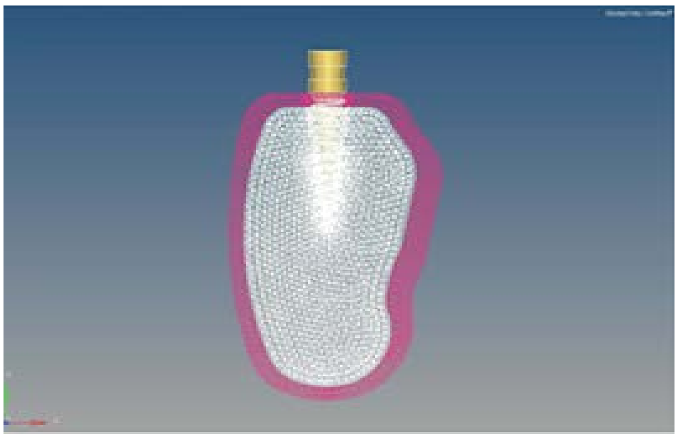

Fig 3: Model after meshing process

This model is then transferred to the software "HYPERMESH13 version 11.0" for the process of meshing. (Fig 3) Meshing divides the entire model into smaller nodes and elements which make a grid called as "mesh". The mesh acts like a spider web, from each node there extends a mesh element to each of the adjacent nodes. The basic idea is to make calculations at only limited (finite) number of points and then interpolate the results for the entire domain.

Any continuous object has infinite degree of freedom (dofs) and practically it is impossible to solve the problem in this format. FEM reduces the dofs from infinite to finite with the help of meshing (nodes and elements) and all the calculations are made at finite number of nodes. This mesh is programmed to contain the material properties (elastic modulus, Poisson's ratio) which define how the structure will react to certain loading conditions i.e., with the incorporation of material properties the structure simulates the normal model.

Load application

Variable forces occur on the mini-screw and bone during orthodontic treatment. When various orthodontic traction force is applied from the key-ridge to the canine, tractional direction slants to the major axis of the mini-implant and bone. FEM was created with mini implant insertion at 90° and to determine the loading effect, 3 force magnitudes (100 g,150 g, and 200 g) and force directions (90° and 120°) to mimic various clinical conditions were investigated. Force direction was defined as the angle between the loading direction and the long axis of the miniscrew, and a force direction of 90° was the force perpendicular to the long axis of the miniscrew. [(6)] Similarly, 3 forces (100 g, 150 g, and 200 g) were applied to the head of the mini-implant or abutment at an angle of 90° and 120° to the bone surface.[(7)] On the application of different loads at 90° and 120° angulation, stress, strain and deformation values were calculated on the bone.

The analysis is done using the software "ANSYS 13 version 14.5 Creo 3.0". The maximum equivalent Von Mises stresses, strain and deformation were analyzed in all the models at the entire given load at 90° and 120° angle. The stress, strain and deformation were visualized in colour coding ranging from dark blue (minimum) to red (maximum) in the models. All the stress values attained were represented in Megapascals (MPa) and deformation in millimeter (mm). The collected data had been entered in MS Excel followed by the analysis using SPSS Trial version

Results

In the present study the model was subjected to different force levels at different angulations and thus the generated stress, strain and deformation values and patterns on the surrounding bone were observed. They were visualized in color coding ranging from dark blue (minimum) to red (maximum) in the models. All the stress values were presented in Megapascals (MPa).

Effect of force on the bone at 90˚angulation

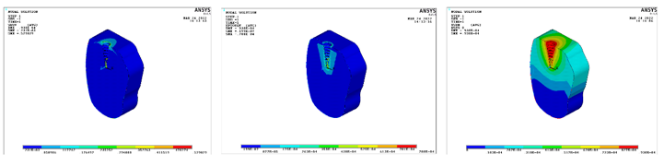

Fig 4: 100 grams (A) Stress (B) Strain (C) Displacement

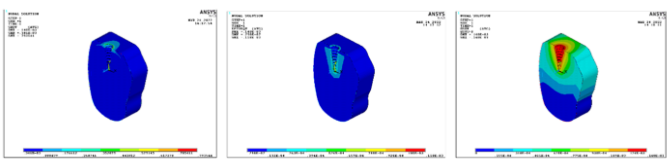

Fig 5: 150 grams (A) Stress (B) Strain (C) Displacement

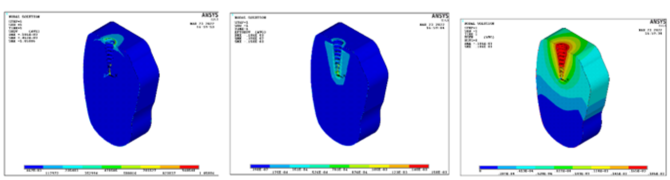

Fig 6: 200 grams(A) Stress (B) Strain (C) Displacement

Table 2: Maximum stress, strain and deformation seen on bone due to the effect of different forces (100 grams, 150 grams and 200 grams) at 90° angulation at different levels.

Straight		Stress(Mpa)	Strain	Deformation(mm)
		Bone	Bone	Bone
100 Grams	Cervical	0.529	0.877e-4	0.930e-4
	Middle	0.058	0.263e-4	0.827e-4
	Apical	0.058	0.788e-4	0.827e-4
150 Grams	Cervical	0.793	0.132e-4	0.140e-3
	Middle	0.088	0.394e-4	0.140e-3
	Apical	0.088	0.118e-3	0.109e-3
200 Grams	Cervical	1.058	0.175e-4	0.186e-3
	Middle	0.117	0.526e-4	0.186e-3
	Apical	0.117	0.158e-3	0.145e-3

At 100 grams of force, maximum amount of stress, strain and deformation was observed in cervical third were 0.529 Mpa, 0.877e-4 and 0.930e-4 respectively. The least amount of stress, strain and deformation was observed in middle third with 0.058Mpa, 0.263e-4 and 0.827e-4 respectively followed by apical third with 0.058Mpa, 0.788e-4 and 0.827e-4 respectively. (Fig.4)

At 150 Grams of force, maximum amount of stress was observed in cervical third with 0.793 Mpa. The least amount of stress was observed in middle third and apical third with 0.088Mpa each. However, the strain was higher in middle third with 0.394e-4. Cervical third and apical had the values of 0.132e-4 and 0.118e-3 respectively. Deformation was similar in cervical, middle and apical third with the values of 0.140e-3, 0.140e-3 and 0.109e-3 respectively. (Fig.5)

At 200 Grams of force, maximum amount of stress was observed in cervical third with 1.058Mpa followed by middle and apical third with 0.117Mpa each. Strain was observed higher in the middle third with 0.526e-4 followed by cervical and apical third with 0.175e-4 and 0.158e-3 respectively. Deformation was similar with values of 0.186e-3, 0.186e-3 and 0.145e-3 in the cervical, middle and apical third. (Fig.6)

Effect of force on the bone at 120˚ angulation

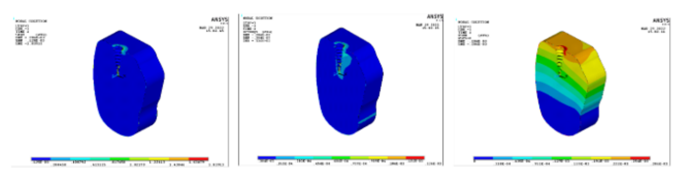

Fig 7: 100 grams (A) Stress (B) Strain (C) Deformation

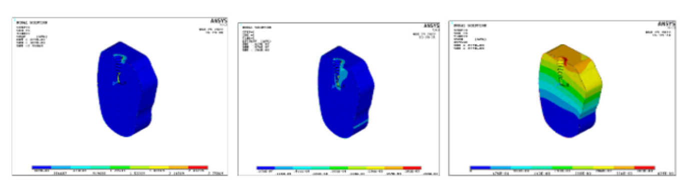

Fig 8: 150 grams (A) Stress (B) Strain (C) Deformation

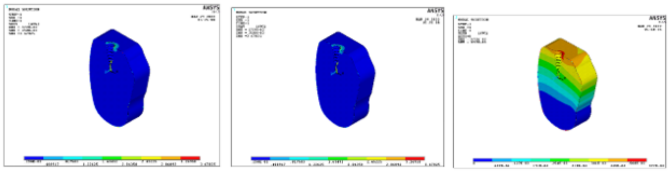

Fig 9: 200 grams (A) Stress (B) Strain (C) Deformation

Table 3: Maximum stress, strain and deformation seen on bone due to the effect of different forces (100 grams, 150 grams and 200 grams) at 120° angulation at different levels.

Straight 120Degree		Stress(Mpa)	Strain	Deformation (mm)
		Bone	Bone	Bone
100 Grams	Cervical	1.839	0.136e-3	0.286e-3
	Middle	0.204	0.454e-4	0.191e-3
	Apical	0.204	0.303e-4	0.159e-3
150 Grams	Cervical	2.759	0.204e-3	0.429e-3
	Middle	0.306	0.608e-4	0.286e-3
	Apical	0.306	0.455e-4	0.238e-3
200 Grams	Cervical	3.678	0.273e-3	0.572e-3
	Middle	0.408	0.909e-4	0.381e-3
	Apical	0.408	0.606e-4	0.318e-3

At 100 grams of force, maximum amount of stress, strain and deformation was observed in cervical third with 1.839 Mpa, 0.136e-3 and 0.286e-3 respectively. The least amount of stress, strain and deformation was observed in apical third with 0.204Mpa, 0.303e-4 and 0.159e-4 respectively followed by middle third with 0.204Mpa, 0.454e-4 and 0.191e-4 respectively. (Fig.7)

At 150 grams of force, maximum amount of stress, strain and deformation was observed in cervical third with 2.759 Mpa, 0.204e-3 and 0.429e-3 respectively. The least amount of stress, strain and deformation was observed in apical third with 0.306Mpa, 0.455e-4 and 0.238e-4 respectively followed by middle third with 0.306Mpa, 0.608e-4 and 0.286e-4 respectively. (Fig.8)

At 200 grams of force maximum amount of stress, strain and deformation was observed in cervical third with 3.678 Mpa, 0.273e-3 and 0.572e-3 respectively. The least amount of stress, strain and deformation was observed in apical third with 0.408Mpa, 0.606e-4 and 0.318e-4 respectively followed by middle third with 0.408Mpa, 0.909e-4 and 0.381e-4 respectively. (Fig.9)

Discussion

The FEM simulated the biomechanical force system that is applied clinically and the response of dentoalveolar system was evaluated. [8] The loading forces applied in this study were within the optimum ranges for the clinical conditions. It is observed that the majority of the mini implants have the ability to stand 100-200 g of horizontal load with ease and the magnitude is sufficient for various tooth movements.[1]

In this study, the orthodontic force levels selected were 150, 200, and 250 g to simulate clinically viable conditions such as individual canine retraction using force of 150 g or en masse retraction using horizontal component of force in the range of 200-250 g.

Buchter L et al, reported in their study that the immediate loading of mini-implants can be performed without any loss of stability. When the load related biomechanics do not exceed an upper loading level, it may even enhance the osseo-integration process.[9]

Since the primary retention of mini-implants is achieved by mechanical means with the bone rather than through osseointegration, the present study evaluates the stress, strain and deformation of the bone surrounding the mini implant. Alveolar cortical bone thickness and density appear to play an important role when planning a mini-implant placement. [10,11]

In the present study, it was observed that for a given load, i.e. 100g, 150g and 200g, the stress, strain and deformation values generated in surrounding bone was least at 90° angulation of force followed by 120° angulation.

This is in accordance with the study by Marimuthu et al[12]. In his study, in 90° implant insertion angle, the maximum stress on bone was observed with an orthodontic force of 60° in maxilla and 120° angulation in mandible.

Thus, the results of the present study indicate that the use of a perpendicular loading force to the mini implant at 90° minimizes stress, strain and deformation rather than 120° and 60° angulation of force.

However, the result of the present study is in contrast with the study by Meher *et al.* [13] who indicated that the use of an angulated loading force to a perpendicularly placed mini-implant minimizes stress.

Whereas Lin *et al.* [14] conducted a study and reported that the orthodontic force direction had no statistically significant effect on stress values in cortical bone. He also found that the insertion angle of the mini-implant significantly influenced the stress values on bone.

Comparing the stress values between the cortical and cancellous bone, Suzuki et al [15] found that the maximum stress of cancellous bone was significantly lower than for cortical bone.

The results of the present study also shows that higher amount of stress, strain and deformation was recorded at the cervical third of the bone surrounding the mini implant rather than middle third and the apical third.

This is in accordance with the study by Clift *et al.* [16] who stated that the stress and strain of the bone gradually reduced along the length of the implant. The contours of von Mises stress and corresponding strain in the results of his study indicated that the load transfer was occurring along the tapered faces and within the recesses along the length of the implant. His study also showed that the highest stress and strain was at the neck of the implant.

The results of the present study is further supported by another study by Vasques *et al* [17]who stated the stress concentration was localized in the cervical margin.

However a photoelastic stress analysis study by Lakha et al [18] showed that higher stresses were encountered in the apical region as compared to cervical region irrespective of the type of load and implant used. The results of the study also stated that oblique load generated highest stress in all the models as compared to axial load.

Borchers et al[19] have reported through Finite-element analysis (FEA) that stress concentrations do occur in the marginal peri-implant bone after lateral or oblique load application.

The present study also showed that von Mises stress values increased with increasing horizontal loading force which is in accordance with the study by Sidhu M et al. [1]

The von Mises stress in the mini implant was mostly present at the neck of the implant at the cervical third close to bone-implant interface. The simulated outcomes, stress, strain and deformation were almost linearly proportional to the force magnitude in this study.

These results were reasonable and predictable, because the material properties in all components were assumed to be isotropic and homogeneous.

Conclusion

The FEM study in the present study tried to accurately evaluate the stress, strain and deformation on the bone surrounding the mini implant; However there is a variation in the clinical response and this will be dependent on many factors, including the host response, mini-screw dimensions, insertion technique, sterilization protocol and type of loading, amongst others.[20] Overloading must be avoided and it is recommended to prevent or minimize the horizontal forces on the cervical level of mini implant for better stability.

The study was done on the basis of assumption that cortical and trabecular bone was isotropic and homogenous. Hence with the current knowledge it is difficult to predict the outcome of treatment with same loading condition over passage of time. Hence it must be used as a reference in assisting clinical judgement.

References

1. Sidhu M, Chugh VK, Dmello K, Mehta A, Chugh A, Tandon P. Evaluation of Stress Pattern Caused by Mini-Implant in Mandibular Alveolar Bone with Different Angulations and Retraction Forces: A Three-Dimensional Finite Element Study. Turk J Orthod. 2020 Jul 18; 33(3):150-156.
2. Pouyafar V, Meshkabadi R, Sadr Haghighi AH, Navid A. Finite element simulation and statistical investigation of an orthodontic mini-implant's stability in a novel screw design. Proceedings of the Institution of Mechanical Engineers, Part H: Journal of Engineering in Medicine. 2021;235(9):1046-1057.
3. 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. American Journal of Orthodontics and Dentofacial Orthopedics. 2013 Feb 1;143(2):182-9.
4. Lu YJ, Chang SH, Ye JT, Ye YS, Yu YS. Finite Element Analysis of Bone Stress around Micro-Implants of Different Diameters and Lengths with Application of a Single or Composite Torque Force. PLoS One. 2015 Dec 14; 10(12):e0144744.
5. Mešić E, Muratović E, Redžepagić-Vražalica L, Pervan N, Muminović AJ, Delić M, Glušac M. Experimental & FEM Analysis of Orthodontic Mini-Implant Design on Primary Stability. Applied Sciences. 2021 Jan;11(12):5461.
6. Liu TC, Chang CH, Wong TY, Liu JK. Finite element analysis of miniscrew implants used for orthodontic anchorage. Am J Orthod Dentofacial Orthop 2012; 141: 468-76.
7. Motoyoshi M, Yano S, Tsuruoka T, Shimizu N. Biomechanical effect of abutment on stability of orthodontic mini-implant. A finite element analysis. Clin. Oral Impl. Res. 16, 2005; 480–485.
8. Jiang L, Kong L, Li T, Gu Z, Hou R, Duan Y. Optimal selections of orthodontic mini-implant diameter and length by biomechanical consideration: A three-dimensional finite element analysis. Adv Eng Software 2009; 40: 1124-30
9. Buchter L, Wiechmann D, Koerdt S, Wiesmann HP, Piffko J, Meyer U. Load-related implant reaction of mini-implants used for orthodontic anchorage. Clin Oral Impl Res 2005; 16: 4.
10. Cassetta M, Sofan AAA, Altieri F, Barbato E. Evaluation of alveolar cortical bone thickness and density for orthodontic mini-implant placement. J Clin Exp Dent. 2013;5(5):e245-52.
11. Chun YS, Lim WH: Bone density at interradicular sites: implications for orthodontic mini-implant placement Orthod Craniofac Res 2009;12:25–32
12. Marimuthu VK, Kumar K, Sadhasivam N, Arasappan R, Jayamurugan A, Rathinasamy R. Finite element analysis of stress and displacement around mini-implant using different insertion angles and various direction of orthodontic force in maxilla and mandible. J Indian OrthodSoc 2015; 49:61-6.
13. Meher AH, Shrivastav SS, Vibhute PJ, Hazarey PV. Deflection and stress distribution around mini-screw implants: a finite element investigation into the effect of cortical bone thickness, force magnitude and direction. J Orthod. 2012 Dec;39(4):249-55.

14. 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; 143: 182-9.

15. Suzuki A, Masuda T, Takahashi I, Deguchi T, Suzuki O, Takano-Yamamoto T. Changes in stress distribution of orthodontic miniscrews and surrounding bone evaluated by 3-dimensional finite element analysis. Am J OrthodDentofacialOrthop. 2011 Dec;140(6):e273-80.

16. Clift SE, Fisher J, Watson CJ. Stress and strain distribution in the bone surrounding a new design of dental implant: a comparison with a threaded Branemark type implant. Proceedings of the Institution of Mechanical Engineers, Part H: Journal of Engineering in Medicine. 1993 Sep;207(3):133-8.

17. Vasquez M, Calao E, Becerra F, Becerra F, Ossa J, Enriquez C, et al. Initial stress difference between sliding and sectional mechanics with an endosseous implant as anchorage: A 3 dimensional finite element analysis. Angle Orthod 2001; 71: 247-56.

18. Lakha TA, Kheur MG, Kheur SM, Patankar AH, Burhanpurwala M. Effect of Implant Design and Thread Geometry on Stress Distribution in the Surrounding Bone-A Photoelastic Stress Analysis. connections. 2016;13:14.

19. Borchers L, Reichart P. Three-dimensional stress distribution around a dental implant at different stages of interface development. Journal of Dental Research. 1983 Feb;62(2):155-9.

20. Park HS, Jeong SH, Kwon OW. Factors affecting the clinical success of screw implants used as orthodontic anchorage. Am J Orthod Dentofacial Orthop 2006; 130: 18-25.