Bharath Ramesh et al /J. Pharm. Sci. & Res. Vol. 13(5), 2021, 302-312

ISSN:0975-1459
Journal of Pharmaceutical Sciences and Research
www.jpsr.pharmainfo.in

Analysis of stress in bone with orthodontic mini-implants during en-masse retraction of maxillary and mandibular anterior teeth: a finite element analysis.

Bharath Ramesh[1], Dilip Srinivasan[2], Sangeetha Duraiswamy[2]

[1] Reader, Ragas Dental College and Hospital,
2/102, East coast road, Uthandi, Chennai, Tamil Nadu- 600119.
[2] Professor, SRM Dental College and Hospital,
Bharathi Salai, Ramapuram,Chennai, Tamil Nadu- 600089.

Abstract:

Introduction: The proper angle of microimplant insertion is important for cortical anchorage, patient safety, and biomechanical control. However, the actual impact of different insertion angulations on stability is unknown.

Aim: To assess the bone and mini-implant of different lengths and diameters during en-masse retraction of maxillary and mandibular anterior teeth using Finite Element Modeling and Simulation

Methods: To perform 3-dimensional finite element analysis, finite element models of a maxilla and a mandible with types D3 and D2 bone quality, and of microimplants with a diameter of 1.3 mm and lengths of 8 and 7 mm were generated. The microimplants were inserted at 30°, 45°, 60°, and 90° to the bone surface. A simulated horizontal orthodontic force of 200 g was applied to the center of the microimplant head, and stress distribution and its magnitude were analyzed with a 3-dimensional finite element analysis program.

Results: The maximum von Mises stresses in the microimplant and the cortical bone decreased as the insertion angle increased. Analysis of the stress distribution in the cortical and cancellous bones showed that the stress was absorbed mostly in the cortical bone, and little was transmitted to the cancellous bone. The maximum von Mises stress was higher in type D3 bone quality than type D2 bone quality.

Conclusion: The bone stress decreased with increasing mini-implant diameter in maxilla and mandible. Length of the mini-implant did not have any considerable effect on stress at implant-bone interface in both maxilla and mandible. Stress in the cancellous bone was considerably less compared to the stress in the cortical bone.

Key Words: En-masse retraction, Implant, Cancellous bone, Cortical bone, Periodontal Ligament, Stress

INTRODUCTION:

Anchorage is the resistance to unwanted tooth movement and the control of anchorage is one of the most critical factors in orthodontic treatment. A goal of any orthodontic treatment is to achieve desired tooth movement with minimum number of undesirable side effects [1] Traditionally, anchorage is reinforced by increasing the number of teeth bilaterally or using extra-oral devices, musculature, and the alveolar processes [2]. However with these traditional methods it is extremely difficult to undertake orthodontic treatment without compromising anchorage in some way [3].

Prevention of undesirable tooth movement in both arches is now possible with the use of orthodontic mini-implants. Orthodontic mini-implant provide the biomechanical advantage that allows more effective and efficient treatment with fewer auxiliaries .It has shown great promise as a simpler and more versatile solution for obtaining absolute anchorage and has increased the envelope of orthodontic treatment [2]. When compared to conventional dental implants used for orthodontic anchorage, the orthodontic mini-implant which is a temporary implant provides many advantages such as; simpler surgical procedure, less trauma during insertion and removal, minimal anatomical limitation, immediate loading after placement and low cost [4,5]

The proper length and thickness of orthodontic mini-implants is important for improved cortical anchorage and better biomechanical control. The effect of different length and thickness of mini-implant on the stress pattern of bone and mini-implants are poorly understood. It is virtually impossible to measure stress accurately around orthodontic mini-implants in vivo. Also, it is difficult to achieve an analytical solution for problems involving complicated geometries such as the maxilla and the mandible, which are exposed to various kinds of loads [6].

Since clinical determination of stress and strain distribution in the bone and mini-implant is not possible, an alternative method has to be used. One such experimental design is the use of the Finite Element Method which provides a way to predict stresses effectively within an object [7].

Finite element analysis provides a mathematical solution for the response of external loads applied to the 3-dimensional (3D) model. It is suitable for simulating complex mechanical system to predict stress in the maxillofacial region [8]. This method is a mathematical method where in the shape of complex geometric objects and their properties are computer constructed and is a highly precise technique used to analyze structural stress. Used in engineering for years, this method uses a computer to solve large number of equations to calculate stress on the basis of physical properties of structures being analyzed [9].

FEM has many advantages over other methods like the photo elastic method, because of the ability to include heterogeneity of tooth material and irregularity of the tooth contour in the model design and the relative ease with which loads can be applied at different directions and magnitudes for a more complete analysis [10].

Finite element analysis has been used in dentistry to investigate a wide range of topics, such as the structure of teeth, biomaterials and restorations, dental implants, and root canals.

In Orthodontics, FEM has been used successfully to model the application of forces to single and multiple tooth systems.[6,10,11] FEM by simulating the real life situation was also used to show that areas of bone remodeling in vitro.12 However, there are limited studies in the literature that have evaluated the effect of different length and width of orthodontic mini implants on stress pattern generated in the alveolar bone. The purpose of this study was to assess the bone and mini-implant of different lengths and diameters during en-masse retraction of maxillary and mandibular anterior teeth using Finite Element Modeling and Simulation. Objectives of the study included 1) to construct a 3 dimensional finite element model of the maxillary and mandibular teeth. 2) to construct 0.022 inch pre-adjusted edgewise bracket system. 3) to simulate the en-masse maxillary and mandibular retraction with five different mini-implant lengths (5, 6, 7, 8and 10mm) and three different widths (1.2, 1.3 and 1.4mm). 4) to analyze the bone and mini-implant stress during en-masse retraction of maxillary and mandibular anterior teeth in all the groups.

MATERIALS AND METHODS:

This study was conducted using 3-dimensional finite element analysis to evaluate and compare the stress distribution of bone around the mini implants during en-masse retraction with sliding mechanics using orthodontic mini-implants of five different lengths and three different widths.

The materials used for this study were: Computer hardware: A HP Workstation XW8200 with an Intel (R) XEON processor, motherboard capacity of 3.40 GHz, 3.50 MB RAM, 80 GB storage space with graphic accelerator. Modeling process: Three dimensional finite element models were created after scanning with computer tomography with slice thickness of 1.5mm for the following structures 1) Maxilla and mandible with dentition, 2) Periodontal ligament, 3) A standard edgewise bracket Roth prescription, slot size, 0.022 x 0.025 inch (3M Unitek, Monrovia, Calif), 4) Stainless steel archwire 0.019 x 0.025 inch (Roth Tru-arch forms, medium size; Ormco, Orange, Calif), 5) Stainless steel hook, 6) Nickel titanium closed-coil spring, 7) Absoanchor mini-implants (Dentos, Taegu, Korea).

These scanned images were viewed with dental EZIDICOM (National Electrical Manufacturers Association, NEMA) and these images were then copied to AUTOCAD to trace the images and the traces were arranged in complete set to make for a single unit using modeling software PRO/ENGINEER WILD FIRE 2.0

The assembly of a single unit was transferred to modeling software PRO/ENGINEER Wildfire 2.0 version (Parametric Technology, Needham, Mass) to create volumes and areas for all the teeth, alveolar bone, periodontal ligament as a solid complex, thus obtaining a geometric model.

The process of meshing was carried using a software ALTAIR HYPERMESH 7.0 version. This is a preprocessor used for preprocessing which includes meshing and applying specific boundary conditions.

The complete model from PRO/E was imported into the HYPERMESH software as an assembly and all the independent parts, via, tooth, bone, wire, brackets & PDL have been meshed with specific element types based on the geometry as given below.

All these components were individually modeled and then assembled to create 3D finite element model of maxilla and mandible depicting en-masse retraction of 6 anterior teeth.

The entire assembly was then exported for analysis with ANSYS Workbench (version 11.0; ANSYS, Canonsburg, Pa) with a bidirectional understandable translated system called initial graphics exchange specification.

CONSTRUCTION OF THE TEETH

A 3D finite element model of each tooth was constructed with reference to the method of Wheeler.71 the maxillary and mandibular dentitions were established according to the normal arch shapes of Roth (Tru-arch forms, medium size; Ormco, Orange, Calif). The teeth were aligned with reference to the facial axis point of Andrews[72] The labiolingual and buccolingual inclinations of the teeth were simulated with reference to studies by Kim et al.73

CONSTRUCTION OF BRACKET AND ARCHWIRE

Three-dimensional finite element models of 0.022 x 0.025-inch standard preadjusted edgewise brackets (3M Unitek, Monrovia, Calif) were modeled and positioned to the crown so that the facial axis point was at the center of the bracket slot.

Three-dimensional finite element models of the archwires (0.019 x 0.025 inch) were designed according to Roth's normal arch.

CONSTUCTION OF THE PERIODONDAL LIGAMENT

The 3D finite element models of the periodontal ligament were constructed with reference to the studies by Kronfeld and coolidge the thickness of the periodontal ligament was considered to be 0.25 mm evenly, although periodontal ligament thickness is different according to age, position, and individual variations [74,75] The 3D finite element models of the alveolar bone were fabricated to fit the teeth and the periodontal ligament.

CONSTRUCTION AND POSITION OF HOOK

Four hooks were simulated and attached perpendicular to the archwire distal to lateral incisor of maxillary and mandibular arch.

CONSTRUCTION AND FORCE GIVEN BY THE CLOSED-COIL SPRING

Nickel titanium closed-coil springs were simulated to deliver the force between the mini-implant and the hook. They generated 200 grams of force to the anterior teeth for en-masse retraction.

CONSTRUCTION OF MINI-IMPLANT

Three-dimensional finite element models of 15 types of small head, self drilling, tapered AbsoAnchor mini-implants of dimension 1.2x5, 1.2x6, 1.2x7, 1.2x8, 1.2x10, 1.3x5, 1.3x6, 1.3x7, 1.3x8, 1.3x10, 1.4x5, 1.4x6, 1.4x7, 1.4x8 and 1.4x10mm using the dimensions and measurements obtained from the AbsoAnchor company were constructed.

POSITIONING OF THE MINI-IMPLANT

The mini-implants were placed 6 mm from the alveolar crest in the interradicular space between the first molar and the second premolar in the maxilla, and 11 mm from the alveolar crest in the interradicular space between the first molar and the second premolar in the mandible with reference to the studies by Poggio et al.[45]

INSERTION ANGLE OF MINI-IMPLANT

The mini-implants were inserted at a constant 900 angulations to the bone surface reference to a finite element study. which concluded from the results that the least amount of stress were noticed with an insertion angle of 900, thus showing that mini-implants should be placed as perpendicular to the bone as possible for better stability [6].

MATERIAL PROPERTIES

The material properties were assigned to the various structures such as the alveolar bone, tooth, periodontal ligament etc in the Finite Element model. The material properties assigned are in confirmation with the data available in previous studies. The material properties were calculated according to the methods of Reimann et al and Vollmer et al. [47,76]

Type 3 (D3) bone quality was present in the posterior maxilla with a thin layer (1 mm) of cortical bone surrounding a core of dense trabecular bone of favorable strength. Type 2 (D2) bone quality was present in the posterior mandible with a thick layer (2 mm) of compact bone surrounding a core of dense trabecular bone according to the study by Lekholm et al. [77]

In this study bone block of dimension 8 x 14 x 10 mm (height, width, and depth) was simulated to simplify the model and reduce the time for analysis. These bone blocks were modeled to represent the sections of the maxilla and the mandible in the interradicular spaces between the first molar and the second premolar where the mini-implant of various length and diameter was inserted at 900 angulations.The section of bone block was simulated with 1mm outer cortical bone thickness in the insertion areas of the maxilla and 2 mm outer cortical bone thickness of the mandible, 15 models for the maxilla and the mandible bone block were simulated for mini implant with various length and diameter. The retraction force of 200 grams was loaded mesiodistal to the center of mini-implants with closed-coil springs. The stress distribution and its magnitude were analyzed by ANSYS Workbench, a 3D finite element analysis program. An assessment of the stress on the bone elements was performed by using von Mises equivalent stress.

All charts and tables were created using Microsoft Excel 2007.Statistical significance analyses were not carried out since results of FEA are individual values without any statistical spread [7].

RESULTS:

The results of this study were viewed in the post-processor of analysis software. The stress distribution was evaluated according to the stress hypothesis of von Mises[78]. The load was described as being the three principal stresses (PS) that determine the general tensional state of a body, namely: σ1 maximum principal stress; σ2 intermediate principal stress and σ3 minimal principal stress. In some cases, the occurrence of principal stresses could mask the tensional state of the body at a certain point. Thus in these cases it is easier to use only a single number, i.e. that of Von Mises stress σe , which describes the proximity of the end of the elastic behavior at this point [79]. These principal stresses can be converted to von Mises stress, and the corresponding shear stress that represents the general effective stress in a material [80] This is a function of the principal stress in an element and is a common way of representing the stresses, since von Mises stresses include components of 6 stresses [81]

For an overview of the stress distribution, a color scale with 9 stress values served to evaluate quantitatively the stress and displacement distributions in mandible [82]. The scale for stress runs from 0 MPa (blue) to the highest stress values (red). Red indicates areas with the highest stress, and blue indicates areas with the lowest stress [6]. In the color scale diagram showing stress distribution, the values are in Newton per square millimeter or megapascals.

The maximum von Mises stress for the mini-implant, cortical bone and cancellous bone were tabulated, these are shown in (Table 2). Maximum von Mises stress on the mini-implant were highly concentrated in the head and neck of the mini-implant, the contact point between implant thread and cortical bone, and the cortical bone surrounding the mini-implant.

The stress induced on cancellous bone is much less than that on the cortical bone irrespective of the length and diameter of the mini-implant6. Thus there was very little stress distribution seen at the mini-implant and cancellous bone interface. The comparison of the maximum von Mises stress values are depicted in (Graph 1) (Table 1).

The maximum von Mises stress in the implant-cortical bone interface gradually decreased with increasing the diameter from 1.2 mm to 1.4 mm in both maxilla and mandible, this has been depicted in (Graph 1- Graph 5, Graph 9-Graph 13). Thus, maximum von Mises stresses observed in the implant-cortical bone interface were highest when the mini-implant diameter was 1.2 mm and stress was lowest when the mini-implant diameter was 1.4 mm.

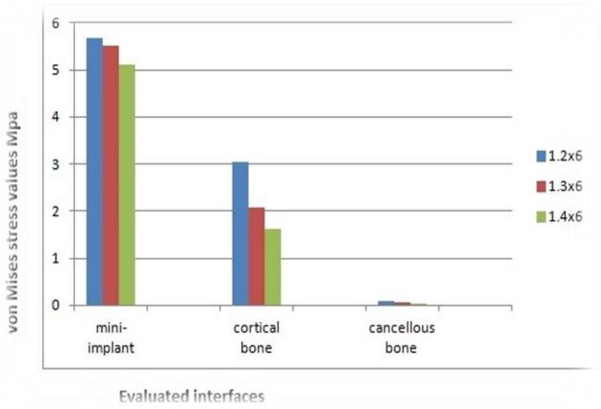
Table 1.Maximum von Mises stress values induced by mini-implant of various lengths and diameters in Mandible

Model	Bone model	Implant (width xlength)	Maximum von Mises stress values		
			Mini-implant (Mpa)	Cortical bone (Mpa)	Cancellous bone (Mpa)
1	Maxilla	1.2 x 5	**3.796**	3.223	.0871
2	Maxilla	1.2 x 6	5.664	3.046	.0827
3	Maxilla	1.2 x 7	5.871	2.946	.0751
4	Maxilla	1.2 x 8	5.343	2.889	.0707
5	Maxilla	1.2 x 10	5.002	2.927	.0853
6	Maxilla	1.3 x 5	5.265	2.097	.0614
7	Maxilla	1.3 x 6	5.513	2.087	.0699
8	Maxilla	1.3 x 7	5.294	2.095	.0512
9	Maxilla	1.3 x 8	5.124	2.106	.0532
10	Maxilla	1.3 x 10	5.547	2.061	.0703
11	Maxilla	1.4 x 5	5.157	1.622	.0853
12	Maxilla	1.4 x 6	5.11	1.619	.0486
13	Maxilla	1.4 x 7	5.98	1.501	.0512
14	Maxilla	1.4 x 8	5.293	1.571	.0424
15	Maxilla	1.4 x 10	5.294	1.478	.0462

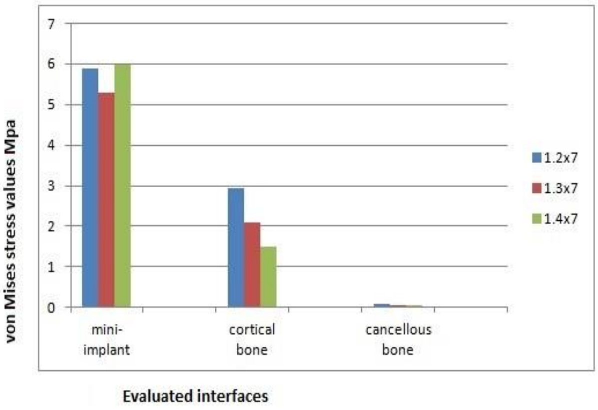
Table 2. Maximum von Mises stress values induced by mini-implants of various lengths and diameters in Maxilla.

Model	Bone model	Implant (width xlength)	Maximum von Mises stress values		
			Mini-implant (Mpa)	Cortical bone (Mpa)	Cancellous bone (Mpa)
1	Mandible	1.2 x 5	5.233	3.546	.0319
2	Mandible	1.2 x 6	5.208	3.385	.0373
3	Mandible	1.2 x 7	5.245	3.487	.0368
4	Mandible	1.2 x 8	5.27	3.498	.0298
5	Mandible	1.2 x 10	5.097	3.558	.0332
6	Mandible	1.3 x 5	5.627	2.716	.0288
7	Mandible	1.3 x 6	5.385	2.642	.0263
8	Mandible	1.3 x 7	5.290	2.586	.0233
9	Mandible	1.3 x 8	5.321	2.533	.0297
10	Mandible	1.3 x 10	5.325	2.696	.0277
11	Mandible	1.4 x 5	5.726	1.497	.0192
12	Mandible	1.4 x 6	5.554	1.382	.0271
13	Mandible	1.4 x 7	5.666	1.399	.0235
14	Mandible	1.4 x 8	5.774	1.298	.0122
15	Mandible	1.4 x 10	6.296	1.309	.0213

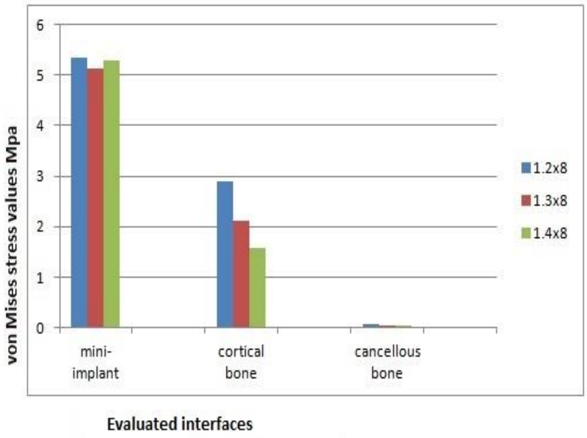
Graph 1.Maximum von Mises stress values induced by mini-implant of three different 0diameters with 5mm length in maxilla



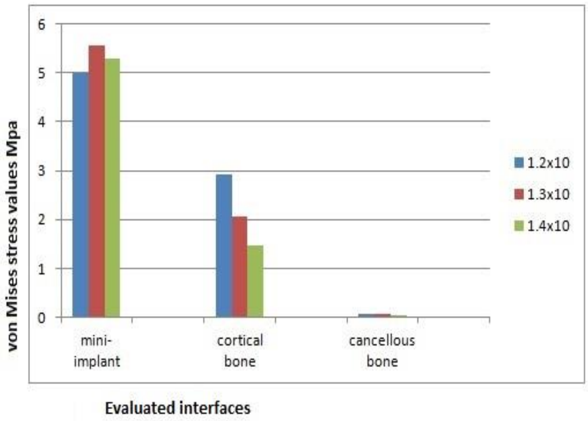
Graph 2.Maximum von Mises stress values induced by mini-implant of three different diameters with 6mm length in maxilla



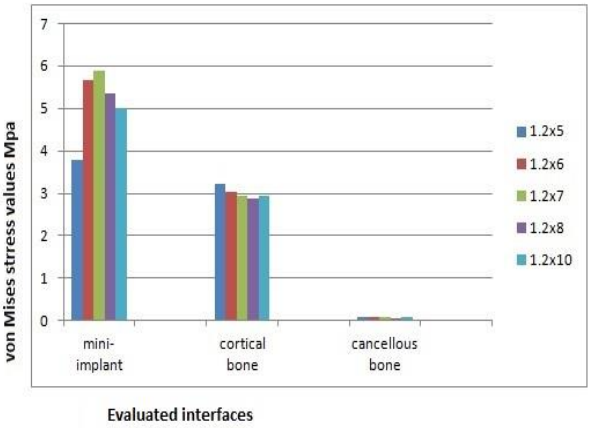
Graph 3.Maximum von Mises stress values induced by mini-implant of three different diameters with 7mm length in maxilla



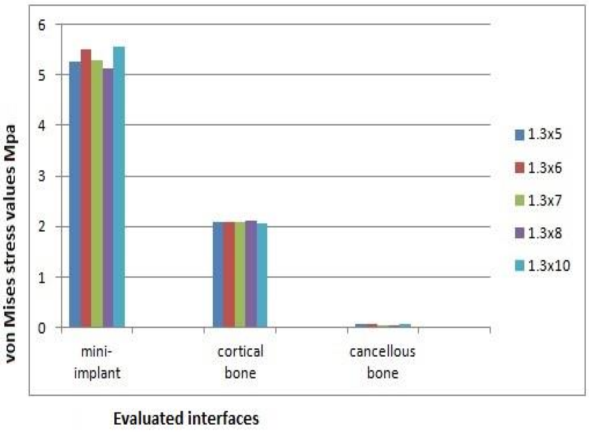
Graph 4.Maximum von Mises stress values induced by mini-implant of three different diameters with 8mm length in maxilla



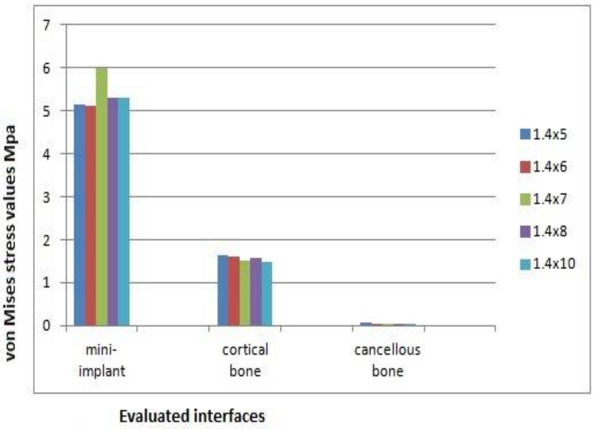
Graph 5.Maximum von Mises stress values induced by mini-implant of three different diameters with 10mm length in maxilla



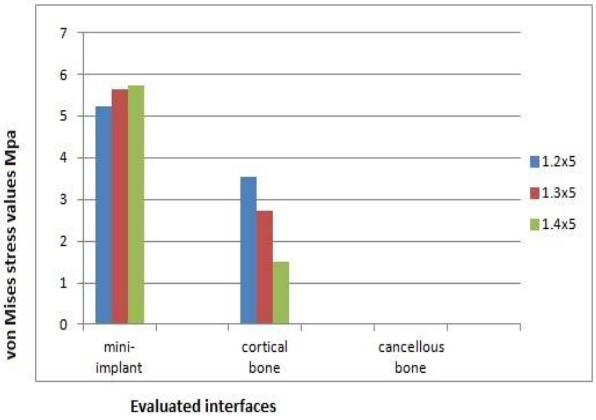
Graph 6.Maximum von Mises stress values induced by mini-implant of five different lengths with 1.2mm diameter in maxilla



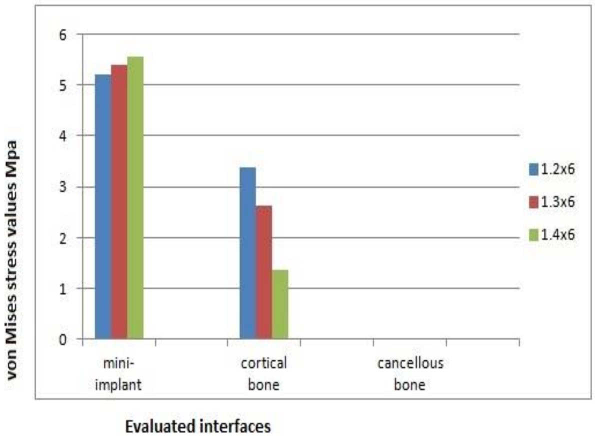
Graph 7.Maximum von Mises stress values induced by mini-implant of five different lengths with 1.3mm diameter in maxilla



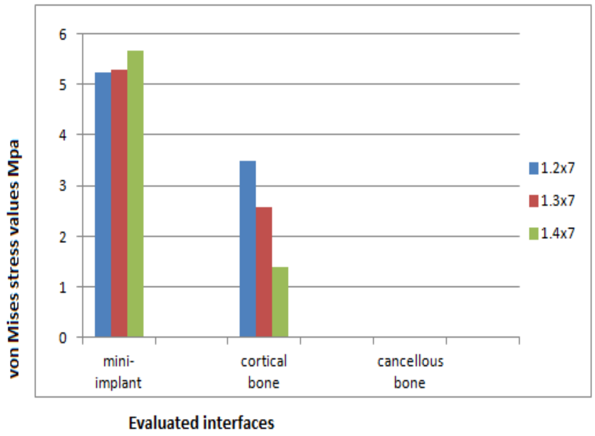
Graph 8.Maximum von Mises stress values induced by mini-implant of five different lengths with 1.4mm diameter in maxilla



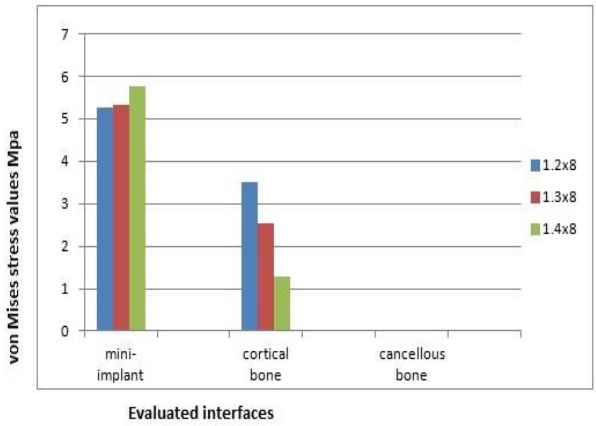
Graph 9.Maximum von Mises stress values induced by mini-implant of three different diameters with 5mm length in mandible



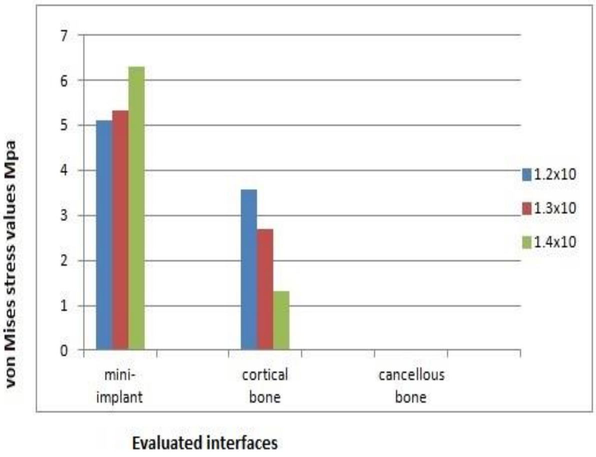
Graph 10.Maximum von Mises stress values induced by mini-implant of three different diameters with 6mm length in mandible



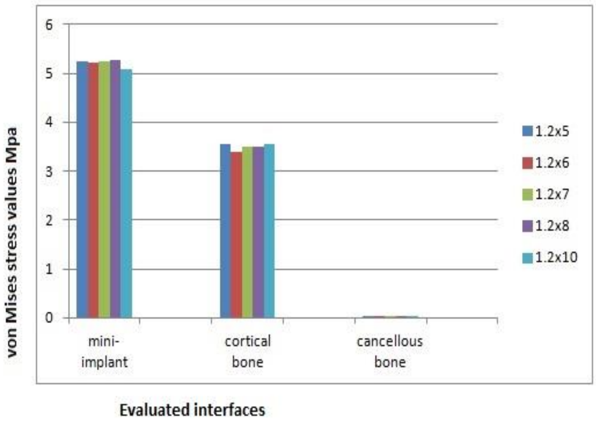
Graph 11.Maximum von Mises stress values induced by mini-implant of three different diameters with 7mm length in mandible



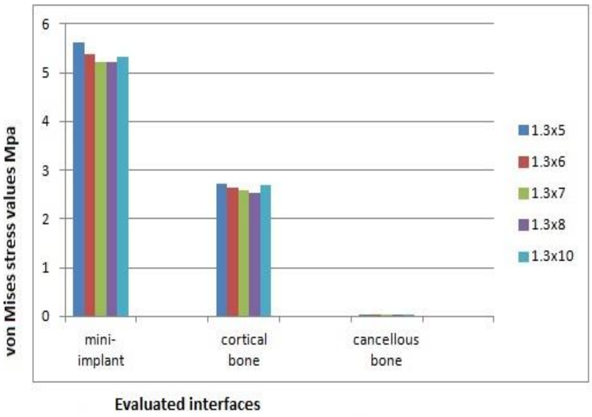
Graph 12.Maximum von Mises stress values induced by mini-implant of three different diameters with 8mm length in mandible



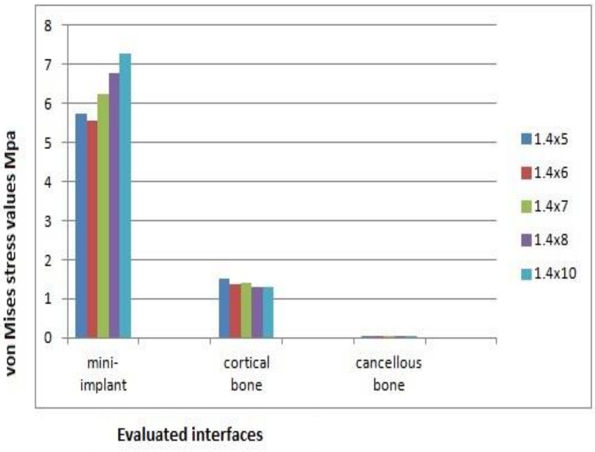
Graph 13.Maximum von Mises stress values induced by mini-implant of three different diameters with 10mm length in mandible



Graph 14.Maximum von Mises stress values induced by mini-implant of five different lengths with 1.2mm diameter in mandible



Graph 15.Maximum von Mises stress values induced by mini-implant of five different lengths with 1.3mm diameter in mandible



Although cancellous bone stress values were very minimal it correlated well to the cortical bone stress at the implant-bone interface and none of the peak stress within cortical bone exceeded their respective ultimate strength limits.

By changing the mini-implant length did not have a considerable effect on maximum von Mises stress at implant-cortical bone interface or implant-cancellous bone interface in both maxilla and mandible. (Graph 6-Graph 8, Graph 14-Table 1)

DISCUSSION:

In orthodontic clinical practice, proper anchorage is a crucial factor for a successful treatment outcome. In recent years mini implants with its absolute anchorage have become the cornerstone of orthodontics. The length and thickness of mini-implant is important for primary stability and clinical success.

Clinically, stress is an important factor in mini-implant stability as increased stress might draw more cytokines, macrophages, and inflammatory mediators to the implant site, possibly resulting in a higher risk of mini-implant failure through loss of primary stability. Biomechanical stresses and strains at the bone-implant interface have been attributed to imlant failure in most instances, resulting in peri-implant inflammation that can lead to bone loss [83]

The concept of finite element analysis originated during 1940’s to study stresses in complex airframe structures. Dr. Ray Cough in 1960 coined the term “finite element”. FEM is a numerical method of analysis, this method schematically divides any object under study into finite number of smaller subunits called elements for analyzing its physical behavior .Each element can be geometric shape (i.e, hexahedron, tetrahedron, triangle, square etc)

Finite element method utilizing the known data of physical properties of the object calculates the deformation of each element under the application of a known load. Thus sum of deformation of the entire element is the deformation of the entire structure. The FEM is a valid and noninvasive method that provides useful results to predict various parameters of the complex biomechanical behavior including bone [84].

Finite element model provides the orthodontist with quantitative data that increases the understanding of the physiologic reactions that occur after force application and may yield an improved understanding of the reactions and interactions of individual tissues and the greatest strength of the finite element model is that it can be magnified nearly infinitely both in terms of the actual volumetric construction itself and the mathematical variability of its material parameters [9,85]

As it is difficult to determine the underlying biomechanical mechanisms for mini-implant applications through an experimental approach because of the limited measureable mechanical index, imprecise parameter control, the large variations among samples, and it is virtually impossible to measure stress in vivo, an alternative method like finite element analysis provides a solution for the response of the 3-dimensional(3D) structures to the applied external loads under certain boundary conditions. It appears to be suitable for simulating complex mechanical stress situations in the maxillofacial region [8,70].

The von Mises stress can be used to predict failure according to the von Mises yield criterion, which states that yielding of a material occurs when the von Mises stress exceeds the yield strength in tension [86]. The von Mises yield criterion applies best to ductile materials such as metals. However, for brittle materials such as bone, the maximum principal stress criterion is commonly used instead of the von Mises yield criterion. The maximum principal stress criterion states that failure occurs when the maximum principal stress reaches either the ultimate tensile strength or the ultimate compressive strength [87].

The recent popularity of mini-implants has lead to its use in various clinical situations. Mini implants are available in a wide choice of length and diameter, but the effect of different length and diameter of mini-implant on the stress pattern of bone and mini-implant are poorly understood and limited studies have been published in respect to this topic. So this study was undertaken to access the stress distribution in the bone and mini-implants of different lengths and diameters during en-masse retraction of maxillary and mandibular anterior teeth using finite element analysis.

In this study, three dimensional finite element models of maxilla, mandible, dentition and periodontal ligament were created by using computer tomography (CT) with slice thickness of 1.5mm. The first premolars were excluded from the model to mimic extraction. Standard preadjusted edgewise brackets (Roth prescription brackets of 0.022 x 0.025-inch, 3M Unitek, Monrovia, Calif) with were generated and placed to the crown so that the facial axis point was at the cenater of the bracket slot and three-dimensional finite element models of the archwires (0.019 x 0.022 inch) were designed according to Roth’s normal arch. Four hooks were simulated and attached perpendicular to the archwire distal to lateral incisor of maxillary and mandibular arch.

Three-dimensional finite element models of 15 types of small head, self drilling, tapered AbsoAnchor mini-implants(1.2x5, 1.2x6, 1.2x7, 1.2x8, 1.2x10, 1.3x5, 1.3x6, 1.3x7, 1.3x8, 1.3x10, 1.4x5, 1.4x6, 1.4x7, 1.4x8, 1.4x10mm) using the dimensions and measurements obtained from the AbsoAnchor company were constructed and placed 6 mm from the alveolar crest in the inter-radicular space between the first molar and the second premolar in the maxilla, and 11 mm from the alveolar crest in the inter-radicular space between the first molar and the second premolar in the mandible with reference to the studies by Poggio et al [45].

In this study a self drilling mini-implant was preferred, because when compared with self tapping mini-implants the placement of self drilling mini-implant is simple, less time consuming, can avoid thermal damage, no risk of drill fracture and also because this system enhances primary stability by compressing bones during implantation [88]. Erma Quraishi et al from their study showed that non-tapered mini-implant fractured at significant higher torque values compared to tapered design, thus tapered design was used in this study [86]

The simulated mini-implants were inserted at a constant 900 angulations to the bone surface with reference to a finite element study, which concluded from the results that least amount of stress were noticed with insertion angle of 900, thus showing that mini-implants should be placed as perpendicular to the bone as much as possible for better stability.6 The material properties were assigned to the various structures such as the alveolar bone, tooth, periodontal ligament according to the methods of Reimann et al and Vollmer et al. [47,76]

Type 3 (D3) bone quality was assigned for the posterior maxilla, with a thin layer (1 mm) of cortical bone surrounding a core of dense trabecular bone of favorable strength. Type 2 (D2) bone quality was assigned for the posterior mandible with a thick layer (2 mm) of compact bone surrounding a core of dense trabecular bone, this was done according to the study by Leholm et al.[77]. Motoyoshi et al from his studies conclude that the minimum cortical bone thickness to ensure mini-implant stability was 1mm [62].

To simplify the model and reduce the time for analysis, a bone block was modeled with dimensions of 8 x 14 x 10 mm (height, width, and depth) for this study. These bone blocks represented the sections of the maxilla and the mandible in the inter-radicular spaces between the first molar and the second premolar where the mini-implant of various length and diameter was inserted at 900 angulations. However these simplifications should affect the quantitative values of the simulations, not the underlying mechanical mechanism [70].

Nickel titanium closed-coil spring was simulated to deliver the 200 grams of force between the mini-implant and the hook. The simulated models were then used to evaluate and compare the stress distribution of implant-bone interface during en-masse retraction with sliding mechanics with ANSYS Workbench, three dimensional finite element method and the assessment of the stress on the bone elements was performed by using von Misses equivalent stress.

The results showed that maximum von Mises stress in the implant-cortical bone interface gradually decreased with increasing the diameter from 1.2 mm to 1.4 mm in both maxilla and mandible. These results correlate well with a study done by Hyo-Sang and others showing that the mini-implant diameter has a major influence on mini-implant stability [52,7,29,55,61,89].

Ramzi and others reported from his finite element study that infra-bony length did not affect the stresses within peri-implant cortical bone but increasing the extra-bony head length of the mini-implants caused an increase in the stresses in bone and may therefore compromise the stability, thus clinically he recommends using a mini-implant with larger diameter in such circumstances [7].

The study also showed that there was not any considerable change on the maximum von Mises stress at implant-cortical bone interface or implant-cancellous bone interface in both maxilla and mandible for all the mini-implant lengths studied. This result was similar to studies done by Sung and others which investigated the importance of mini-implant length on stability of mini-implant and concluded that it does not significantly affect the mini-implant stability [28,7,46,90,91,92]. However the results of this study was not in agreement with studies done by Lim and others [12,58,62]

The study results also showed that there was very little stress distribution seen at the mini-implant and cancellous bone interface. Thus showing that stress induced on cancellous bone is much less when compared to cortical bone irrespective of the dimensions of the mini-implant. Similar results were found by Gracco and Dalstra indicating that most of the stress placed on bone by the mini-implant was absorbed by the cortical bone and therefore may be an important factor for mini-implant stability [60,95].

The patterns of stress distribution of mini-implant under loading of 200 gms of force showed that the area of stress distribution were around the head and neck of the mini-implant, the cortical bone and the upper third of the trabacular bone. The area of maximum stress were concentrated around the point of force application i.e the neck region of the mini-implant. These stress concentration gradually decreased from the neck towards the apex region of the mini-implant. This was probably because the greatest resistance is exerted at the mini-implant entrance into the cortical and cancellous bones. This result obtained correlated well with other similar studies.[6,68,96].

From this study it can be concluded that; the increase in diameter of the mini implant reduces the stress pattern. Therefore, larger diameter of mini-implant should be used wherever possible within the anatomical limitations of the situation. Secondly, the length of the mini-implant had no effect on the cortical bone stress and therefore the selection of the length should be based on factors like primary stability, purpose of use, etc., rather than stress.

Thirdly, there was minimal stress on the cancellous bone which indicate that cancellous bone plays a secondary role to cortical bone in mini-implant placement.

Although finite element analysis is a useful technique, it has few limitations like any other theoretical model of a biological system, which include some basic assumptions such as material properties of the structures is understood to be nonlinearly elastic and anisotropic, but for the purpose of simplicity and lack of scientific quantitative data on bone behavior material properties, the present study assumed that all materials are linear, homogenous and isotropic.

Another limitation of finite element study evaluating stress is that its model neglects the stress produced by the insertion of the screw and considers only the stresses produced by horizontal and torsional loads. Mano from his study concluded that bony tissue behaves as a viscoelastic material, which results in relaxation in the stress fields generated by implant insertion thus implying that stress produced by insertion of screw are less significant [97].

The mini implant-bone interface was simulated to have full contact, which is not the case in real-life situation. Nevertheless, a full contact was assumed for simplicity and to represent the best possible relationship between bone and the implant.

Analytical results of the finite element model depend mainly on the models developed, so they must be constructed to be equivalent to the real object in various aspects. For a more accurate model, more nodes would be needed [6]. However; this will cause a corresponding increase in the number of subsequent computational operations, making the process lengthy and more complex. Hence, it's practical that the number of elements are optimized in the areas of large stress, and kept to bare minimum in the other areas.

The material properties and the geometry of the model differs from person to person and the stress distribution patterns simulated also might differ, depending on the materials and properties assigned to each model used in the study.

The models used in this study were done to reflect the reality as much as possible with respect to their limitations, but the simulation of heterogeneity and anisotropy of bone and the properties of the implant surface must be considered in future studies for evaluating varying force level, implant placement level, shape of the mini-implant, etc.

CONCLUSION:

In spite of the limitations of the study, the results can definitely be used as a guideline for selection of the type of mini-implants and thereby increasing the likelihood of their success in any given clinical scenario.

Within the limitations of this study, the following conclusions were drawn from the results obtained, The bone stress decreased with increasing mini-implant diameter in maxilla and mandible. Length of the mini-implant did not have any considerable effect on stress at implant-bone interface in both maxilla and mandible. Stress in the cancellous bone was considerably less compared to the stress in the cortical bone.

Larger diameter mini-implants tend to produce lesser stress on the cortical bone and varying the length of the mini-implant does not have an effect on stress values. Most of the stress placed on bone by the mini-implant was absorbed by the cortical bone thus cancellous bone may also not be a major factor in mini-implant stability. Further clinical studies are needed to confirm these findings.

REFERENCES:

1. Proffit W.R, Fields H.W, Sarver D.M. contemporary orthodontics. ed 2 .St. Louis; Mosby-yearbook. 1993: 307
2. Jong Suk Lee, Jung Kook Kim, Young-Chel Park, Robert L Vanarsdall, Jr. Applications of Orthodontic Mini-Implants. Quintessence Publishing Co, Inc. 2007: 1
3. Jae-Hyun Sung, Hee-Moon Kyang, Seong-Min Bae, Hyo-Sang Park, Oh-Won Kwon, James A. McNamara jr. Microimplants in orthodontics. 2006: 7
4. Park HS, Kwon TG. Sliding mechanics with microscrew implant anchorage. Angle Orthod 2004;74(5):703-10
5. Cope JB. Temporary anchorage devices in orthodontics: a paradigm shift. Semin Orthod 2005;11(1):3-9
6. Issa Fathima Jasmine. M, A. Arif Yezdani, Faisal Tajir, Murali Venud .R. Analysis of stress in bone and microimplants during en-masse retraction of maxillary and mandibular anterior teeth with different insertion angulations: A 3 dimensional finite element analysis study. Am J Orthod Dentofacial Orthop 2012;141(1):71-80
7. Ramzi Duaibisa, Budi Kusnotob, Raghu Natarajanc, Linping Zhaod, Carla Evanse. Factors affecting stresses in cortical bone around miniscrew implants A three-dimensional finite element study. Angle Orthod. 2012;82(5):875–880
8. Basciftci FA, korkmaz HH, Usumez S, Eraslan O. Biomechanical evaluation of Chincup Treatment with various force Vector. Am J Orthod Dentofacial Orthop. 2008;134(6):773-81
9. Vishal Seth, Prasanth Kamath, Venkatesh M.J. A marvel of modern technology: Finite Element Model. Virtual Journal of Orthodontics.2010;8.4(2):1-5
10. Sagar S Padmawar, Anup Belludi, Amit Bhardwaj, Vinay Vadvadgi,Rajiv Saini Study of Stress Distribution in Maxillary Anterior Region during True Intrusion of Maxillary Incisors using Finite Element Methodology. Int J Experiment Dent Sci 2012;1(2):89-92
11. Cobo J, Sicilia A, Arguelles J, Suarez D, Vijande M. Initial stress induced in periodontal tissue with diverse degrees of bone loss by an orthodontic force: tridimensional analysis by means of the finite element method. Am J Orthod Dentofacial Orthop. 1993; 104(5):448-54
12. Lim SA, Cha JY, Hwang CJ. Insertion torque of orthodontic miniscrews according to changes in shape, diameter and length. Angle Orthod. 2008;78(2):234–240
13. K.R. Williams, J.T. Edmundson. Orthodontic tooth movement analysed by the Finite Element Method. Biomaterials.1984;5(6):347-351
14. 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;92(6):499-505
15. Inoue Y. Biomechanical study on orthodontic tooth movement by means of numerical simulation- Effects of principal stresses in periodontal membrane. Osaka Daigaku Shigaku Zasshi. 1989; 34(2):306-21
16. Pete G Fotos, Constantine C.Spyrakos, Dennis O.Benard. Orthodontic forces generated by a simulated archwire appliance evaluated by the Finite Element Method. Angle Orthod. 1990; 60(4):277-82
17. Andersen KL, Mortensen HT, Pedersen EH, Melsen B. Determination of stress levels and profiles in the periodontal ligament by means of an improved three-dimensional finite element model for various types of orthodontic and natural force systems. J Biomed Eng. 1991;13(4):293-303
18. Wilson AN, Middleton J, McGuinness N, Jones M. A finite element study of canine retraction with a palatal spring. Br J Orthod. 1991(3);18:211-8

19. Puente MI, Galban L, Cobo JM. Initial stress differences between tipping and torque movements. A three-dimensional finite element analysis. Eur J Orthod. 1996;18(4):329-39
20. Alcaniz M, Montserrat C, Grau V; Chinesta F, Ramon A, Albalat S. An advanced system for the simulation and planning of orthodontic treatment. Med Image Anal.1998;2(1):61-77
21. Bourauel C, Freudenreich D, Vollmer D, Kobe D, Drescher D, Jager A. Simulation of orthodontic tooth movements- A comparison of numerical models. J Orofac Orthop.1999;60(4):136-51
22. Zufall SW, Kusy RP. Sliding mechanics of coated composite wires and the development of an engineering model for binding.Angle Orthod. 2000;70(1):34-47
23. Jones ML, Hickman J, Middleton J, Knox J, Volp C. A validated finite element method study of orthodontic tooth movement in the human subject. J Orthod. 2001;28(1):29-38
24. Qian H, Chen J, Katona TR. The influence of PDL principal fibers in a 3-dimensional analysis of orthodontic tooth movement. Am J Orthod Dentofacial Orthop.2001;120(3):272-9
25. Rudolph DJ, Willes PMG, Sameshima GT. A finite element model of apical force distribution from orthodontic tooth movement. Angle Orthod. 2001;71(2):127-31
26. Mestrovic S, Slaj M, Rajic P. Finite element method analysis of the tooth movement induced by orthodontic forces. Coll Antropol. 2003;27 Suppl 2:17-21
27. Poppe M, Bourauel C, Jager A. Determination of the elasticity parameters of the human periodontal ligament and the location of the center of resistance of single-rooted teeth a study of autopsy specimens and their conversion into finite element models. J Orofac Orthop. 2002;63(5):358-70
28. Shouichi Miyawaki, Isao Koyama, Masahide Inoue, Katsuaki Mishima, Toshio Sugahara, Teruko Takano-Yamamoto. Factors associated with the stability of titanium screws placed in the posterior region for orthodontic anchorage. Am J Orthod Dentofacial Orthop 2003;124(4):373-8
29. Sung SJ, Baik HS, Moon YS, Yu HS, Cho YS. A comparative evaluation of different compensating curves in the lingual and labial techniques using 3D FEM. Am J Orthod Dentofacial Orthop.2003; 123(4):441-50
30. Bai D, Cheng BH, Luo SJ, Lu T. Three-dimensional finite element analysis of maxillary canine during the tooth translation movement. Sichuan Da Xue Xue Bao Yi Xue Ban.2004;35(3):358-60
31. Chang YI, Shin SJ, Baek SH. Three-dimensional finite element analysis in distal en masse movement of the maxillary dentition with the multiloop edgewise archwire. Eur J Orthod.2004;26(3):339-45
32. 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;19(1):100-6
33. Clement R, Schneider J, Brambs HJ, Wunderlich A, Geiger M, Sander FG. Quasi-automatic 3D finite element model generation for individual single rooted teeth and periodontal ligament. Comput Methods Programs Biomed.2004;73(2):135-44
34. Qian Y, Fan Y, Jiang W, Cheng B. Three-dimensional finite element stress analysis on the periodontal tissue of maxillary canine. Sheng Wu Yi Xue Gong Cheng Xue Za Zhi.2004;21(2):196-9
35. Cattaneo PM, Dalstra M, Melsen B. The finite element method: a tool to study orthodontic tooth movement. J Dent Res.2005;84(5):428-33
36. Gallas MM, Abeleira MT, Fernandez JR, Burguera M. Three-dimensional numerical simulation of dental implants as orthodontic anchorage. Eur J Orthod.2005;27(1):12-6
37. Hayashi K, Delong R, Mizoguchi I. Comparison of the finite helical axis and the rectangular coordinate system in representing orthodontic tooth movement. J Biomech.2005;39(16):2925-33
38. Kojima Y, Fukui H. Numerical simulation of canine retraction by sliding mechanics. Am J Orthod Dentofacial Orthop.2005;127:542-51
39. Motoyoshi M, Yano S, Tsuruoka T, Shimizu N. Biomechanical effect of abutment on stability of orthodontic mini-implant. A finite element analysis. Clin Oral Implants Res.2005;16(4):480-5
40. M. Sevimay F. Turhan, M. A. Kilicxarslan G. Eskitascioglu. Three-dimensional finite element analysis of the effect of different bone quality on stress distribution in an implant-supported crown. J Prosthet Dent.2005;93(3):227-34
41. Wang D, Yan Y, Wang C, Qian Y. Three-dimensional finite element analysis of the biomechanical effects of multiloop edgewise archwire (MEAW). Sheng Wu Yi Xue Gong Cheng Xue Za Zhi.2005; 22(1):86-90
42. Yan YQ, Yan HQ, Cai Z, Wang DM, Qian YF, Wang CT. A primary construction of the 3D finite element model for fixed edgewise appliance. Shanghai Kou Qiang Yi Xue.2005;14(3):301-5
43. Zhang Y, Zhang D, Feng CJ. A three-dimensional finite element analysis for the biomechanical characteristics of orthodontic anchorage micro-implant. Shanghai Kou Qiang Yi Xue.2005;14(3):281-3
44. Hyo-Sang Park, Seong-Hwa Jeong, Oh-Won Kwon. Factors affecting the clinical success of screw implants used as orthodontic anchorage. Am J Orthod Dentofacial Orthop.2006;130(1):18-25
45. Paola Maria Poggioa, Cristina Incorvatib, Stefano Velob, Aldo Carano. ‘‘Safe Zones’’: A Guide for Miniscrew Positioning in the Maxillary and Mandibular Arch. Angle Orthod.2006;76(2):191–197
46. Motoyoshi M, Yoshida T, Ono A, Shimizu N. Effect of cortical bone thickness and implant placement torque on stability of orthodontic mini-implants. Int J Oral Maxillofac Implants.2007;22(5):779-84
47. Reimann S, Keilig L, Jager A, Bourauel C. Reimann S, Keilig L, Jager A, Bourauel C. Biomechanical finite-element investigation of the position of the centre of resistance of the upper incisors. Eur J Orthod.2007;29(3):219-24
48. Shingo Kuroda, Yasuyo Sugawara, Toru Deguchi, Hee-Moon Kyung, Teruko Takano-Yamamoto. Clinical use of miniscrew implants as orthodontic anchorage: Success rates and postoperative discomfort. Am J Orthod Dentofacial Orthop.2007;131(1):9-15
49. Song YY , Cha JY, Hwang CJ. Mechanical characteristics of various orthodontic mini-screws in relation to artificial cortical bone thickness. Angle Orthod.2007;77(6):979-85
50. Wiechmann D, Meyer U, Buchter A. Success rate of mini- and micro implants used for orthodontic anchorage: a prospective clinical study. Clin Oral Implants Res.2007;18(2):263-7
51. Yu HS, Baik HS, Sung SJ, Kim KD, Cho YS. Three-dimensional finite-element analysis of maxillary protraction with and without rapid palatal expansion. Eur J Orthod.2007;29(2):118-25
52. Hyo-Sang Park, Youn-Ju Lee, Seong-Hwa Jeong, Tae-Geon Kwond. Density of the alveolar and basal bones of the maxilla and the mandible. Am J Orthod Dentofacial Orthop.2008;133(1):30-7
53. Moon CH, Lee DG, Lee HS, Im JS, Baek SH. Factors associated with the success rate of orthodontic miniscrews placed in the upper and lower posterior buccal region. Angle Orthod.2008;78(1):101-6
54. Nam-Ki Lee, Dong-Soon Choi, In-San Jang. Factors influencing primary stability of miniplate anchorage: a three-dimensional finite element analysis. Korean J Orthod.2008;38(5):304-313
55. Wilmes B, Ottenstreuer S, Drescher D. Impact of implant design on primary stability of orthodontic mini-implants. J Orofac Orthop.2008;69(1):42-50 56.Wilmes B , Su YY, Drescher D. Insertion angle impact on primary stability of orthodontic mini-implants. Angle Orthod.2008;78(6):1065-70
57. Brinley CL, Behrents R, Kim KB, Condoor S, Kyung HM, Buschang PH. Pitch and longitudinal fluting effects on the primary stability of miniscrew implants. Angle Orthod.2009;79(6):1156-61
58. Chad Morarend, Fang Qian, Steve D. Marshall, Karin A. Southard, Nicole M. Grosland, Teresa A. Morgan, Michelle McManus, Thomas E. Southardh. Effect of screw diameter on orthodontic skeletal anchorage. Am J Orthod Dentofacial Orthop.2009;136(2):224-9
59. Chun YS, Lim WH. Bone density at interradicular sites: implications for orthodontic mini-implant placement. Orthod Craniofac Res.2009;12(1):25-32
60. Gracco A, Cirignaco A, Cozzani M, Boccaccio A, Pappalettere C, Vitale G. Numerical/experimental analysis of the stress field around miniscrews for orthodontic anchorage. Eur J Orthod.2009;31(1):12-20
61. Micah G. Mortensen, Peter H. Buschang, Donald R. Oliver, Hee-Moon Kyung, Rolf G. Behrentse. Stability of immediately loaded 3- and 6-mm miniscrew implants in beagle dogs—a pilot study. Am J Orthod Dentofacial Orthop.2009;136(2):251-9

62. 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;38(1):13-8
63. Yan Chen, Hee Moon Kyung, Wen Ting Zhao, Won Jae Yu. Critical factors for the success of orthodontic mini-implants: A systematic review. Am J Orthod Dentofacial Orthop.2009;135(3):284-91
64. Michael B. Pickard, Paul Dechow, P. Emile Rossouw, Peter H. Buschang. Effects of miniscrew orientation on implant stability and resistance to failure.Am J Orthod Dentofacial Orthop.2010;137(1):91-9
65. Sang-Jin Sung, Gang-Won Jang, Youn-Sic Chun, Yoon-Shik Moon. Effective en-masse retraction design with orthodontic mini-implant anchorage: A finite element analysis. Am J Orthod Dentofacial Orthop.2010;137(5):648-57
66. Shetty P, Hegde AM, Rai K. Finite element method--an effective research tool for dentistry. J Clin Pediatr Dent.2010;34(3):281-5
67. Yukio Kojima, Hisao Fukui . Numeric simulations of en-masse space closure with sliding mechanics. Am J Orthod Dentofacial Orthop.2010;138(6):702.e1-6
68. Holm L, Cunningham SJ, Petrie A, Cousley RR. An in vitro study of factors affecting the primary stability of orthodontic mini-implants. Angle Orthod.2012;82(6):1022-8
69. Shivani Singh, Subraya Mogra, V. Surendra Shetty, Siddarth Shetty, Pramod Philipe: 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;141(3):327-36
70. Te-Chun Liu, Chih-Han Chang, Tung-Yiu Wong, Jia-Kuang Liuc. Finite element analysis of miniscrew implants used for orthodontic anchorage. Am J Orthod Dentofacial Orthop.2012;141(4):468-76
71. Wheeler RC. A textbook of dental anatomy and physiology. Philadelphia:W. B. Saunders.1965:135-266
72. Andrews LF. The six keys to normal occlusion. Am J Orthod.1972;62(3):296-309
73. Kim JS, Jin KH, Hong SJ. A statistical study of clinical crown inclination in Koreans' naturally occuring optimal occlusion. Korean J Orthod.1992;22(3):715-33
74. Kronfeld R. Histologic study of the influence of function on the human periodontal membrane. J Am Dent Assoc.1931;18:1242-74
75. Coolidge ED. The thickness of the human periodontal membrane. J Am Dent Assoc.1937;24:1260-70
76. Vollmer D, Bourauel C, Maier K, Jeager A. Determination of the centre of resistance in an upper human canine and idealized tooth model. Eur J Orthod 1999;21(6):633-48
77. Lekholm U, Zarb GA, Branemark PI, Zarb GA, Albrektsson T. Tissue integrated prostheses: osseointegration in clinical dentistry.Chicago: Quintessence.1985:199-209
78. Erkmen E, Sims¸ek B, Yücel E, Kurt A. Three-dimensional finite element analysis used to compare methods of fixation after sagittal split ramus osteotomy: setback surgery-posterior loading. Br J Oral Maxillofac Surg.2005;43(2):97-104
79. Martin I. Puente, Luciano Galban and Juan M. Cobo. Initial stress differences between tipping and torque movements. A three-dimensional finite element analysis. Eur J Orthod.1996;18(4):329-339
80. Erkmen E, Sims¸ek B, Yücel E, Kurt A. Comparison of different fixation methods following sagittal split ramus osteotomies using three-dimensional finite elements analysis. Part 1: advancement surgery-posterior loading. Int J Oral Maxillofac Surg.2005;34(5):551-8
81. Maurer P, Holweg S, Knoll WD, Schubert J. Study by finite element method of the mechanical stress of selected biodegradable osteosynthesis screws in sagittal ramus osteotomy. Br J Oral Maxillofac Surg.2002;40(1):76-83
82. Hart RT, Hennebel VV, Thongpreda N, Van Buskirk WC, Anderson RC. Modeling the biomechanics of the mandible: a three-dimensional finite element study. J Biomech.1992;25(3):261-86
83. Noble J, Karaiskos NE, Hassard TH, Hechter FJ, Wiltshire WA. Stress on bone from placement and removal of orthodontic miniscrews at different angulations. J Clin Orthod.2009;43(5):332-4
84. Vollmer D, Meyer U, Joos U, Vegh A, Piffko J. Experimental and finite element study of a human mandible. J Craniomaxillofac Surg.2000;28(2):91-6
85. David J. Rudolph, Michael G. Willes, Glenn T. Sameshima. A Finite element model of apical force distribution from orthodontic Tooth Movement. Angle Orthod.2001;71(2):127–131
86. von Mises R. Mechanik der Festen Ko¨ rper im plastisch deformablen Zustand. Go¨ ttin Nachr Math Phys.1913;1:582–592
87. Rankine W. On the stability of loose earth. Phil Trans RSocLond.1857;147:9–27
89. W. Heidemann, H. Terheyden, and K. Louis Gerlach, "Analysis of the osseous/metal interface of drill free screws and self-tapping screws. J Cranio-Maxillofac Surg.2001;29(2):69–74
90. Erma Quraishi, Martyn Sherriff, Dirk Bister. Peak insertion torque values of five mini-implant systems under different insertion loads. J Orthod. 2014;41(2):102-109
91. Nam-ki lee, seung-hak Baek. Effects of the diameter and shape of orthodontic mini-implants on microdamage to the cortical bone. Am J Orthod Dentofacial Orthop.2010;138(1):8.e1-8
92. Wilmes B, Rademacher C, Olthoff G, Drescher D. Parameters affecting primary stability of orthodontic mini-implants. J Orofac Orthop.2006;67(3):162–174
93. Lim HJ, Eun CS, Cho JH, Lee KH, Hwang HS. Factors associated with initial stability of miniscrews for orthodontic treatment. Am J Orthod Dentofacial Orthop. 2009;136(2):236–242
94. Chen YJ, Chang HH, Huang CY, Hung HC, Lai EH, Yao CC. A retrospective analysis of the failure rate of three different orthodontic skeletal anchorage systems. Clin Oral Implants Res.2007;18(6):768–775
95. Dalstra M, Cattaneo PM, Melsen B. Load transfer of miniscrews for orthodontic anchorage. Orthod.2004;1:53–62
96. van Staden R, Guan H, Loo YC, Johnson N, Meredith N. Finite element simulation of dental implantation process using the finite element technique.Clin Oral Implants Res.2008;19(3):303-13
97. Mano JF. Viscoelastic properties of bone: mechanical spectroscopy studies on a chicken model. Mater Sci Eng C.2005;25(2):145-52