

Effects of Mechanical Vibrations on Stress Distribution and Displacement in Bone and Teeth Surrounding Infrazygomatic Screws: An In-vitro Finite Element Analysis Study

RISHIKA SINGLA¹, ARAVIND KUMAR SUBRAMANIAN², SRUTHI HARIKRISHNAN³

ABSTRACT

Introduction: Infrazygomatic Crest (IZC) Mini-Implants (MI) is widely used in orthodontics for stable anchorage. However, the impact of mechanical vibrations on bone stress distribution and implant stability remains poorly understood. The rationale for this study stems from the need to understand how combined mechanical stimuli affect the local biological environment, which is critical for ensuring the safe and effective use of IZC screws. The study seeks to offer insights into the potential implications of vibration-assisted accelerated orthodontics on skeletal anchorage stability. Finite Element Analysis (FEA) provides a valuable non-invasive tool to simulate and analyse these complex interactions.

Aim: To evaluate and compare the stress distribution and displacement in bone and adjacent teeth surrounding IZC screws under static orthodontic loading and under mechanical vibration.

Materials and Methods: An in-vitro FEA was conducted at a Saveetha Dental College, Chennai, from January to March

2024. A digital 3D model of the maxilla with teeth and an infrazygomatic screw was developed using SolidWorks 2021, meshed in HyperMesh 14, and analysed in Abaqus 6.14. Stress and displacement values were compared under static load (250 g) and vibratory conditions (30-120 Hz). No inferential statistical tests were applied, as this was a simulation-based study.

Results: Under static loading (250 g), minimal stress and displacement were observed. Modal analysis revealed natural frequencies beginning at 1048 Hz, ruling out resonance. Frequency response analysis showed a reduction in bone stress (0.43 MPa) and minor increases in tooth displacement compared to static conditions, suggesting safe vibratory effects.

Conclusion: Mechanical vibrations within safe frequency ranges do not induce resonance and may enhance tooth displacement without compromising structural stability. These findings provide insights for optimising IZC screw applications in orthodontics.

Keywords: Bone remodelling, Computer simulation, Dental stress analysis, Orthodontic anchorage procedures, Tooth movement techniques

INTRODUCTION

Orthodontic treatment relies heavily on stable anchorage to achieve precise tooth movement while minimising undesirable outcomes. Skeletal anchorage systems, particularly MIs, have become indispensable in modern orthodontics due to their versatility and ability to provide stable anchorage with minimal dependence on patient compliance [1,2]. These devices are small, are implanted with a relatively simple surgical procedure, and increase the potential for better orthodontic results. Therefore, miniscrews not only free orthodontists from anchorage-demanding cases, but they also enable clinicians to have good control over tooth movement in three-dimensions [3]. Among these, IZC MIs are recognised for their superior biomechanical properties, including optimal cortical bone thickness at the site of IZC, reduced risk of root damage, and capability to support complex tooth movements like en masse maxillary distalisation [4].

Despite their advantages, factors affecting the long-term stability and stress distribution around IZC screws remain areas of active investigation. Mechanical vibrations- either incidental or deliberate- are hypothesised to influence bone remodeling and stress patterns around orthodontic implants [5]. Understanding these effects is critical for improving MI stability and predicting their performance in clinical settings. However, limited research has explored the direct impact of mechanical vibrations on bone-implant interactions, particularly at the IZC site.

Evaluating stress levels in bone and MI during orthodontic procedures presents significant clinical challenges, making it difficult

to interpret the resulting stress patterns accurately. To overcome these limitations, researchers have increasingly relied on FEA, a powerful engineering tool that allows for precise simulation of stress and strain patterns in living structures. The Finite Element Method (FEM) enables non-invasive geometric modeling of complex structures, facilitating detailed stress and strain analysis [6,7]. FEA is an invaluable tool in this context, enabling detailed simulation and visualisation of biomechanical behaviour under controlled conditions. FEA has been extensively applied in orthodontic research to study stress distribution, displacement, and the impact of various factors on MI stability and efficacy [8]. Its ability to model complex structures, such as the maxilla, and assess localised bone stress makes it ideal for investigating the dynamic effects of mechanical vibrations on IZC screws.

One key metric used in FEM studies is the Von Mises stress, which helps predict whether a material will deform or fracture under complex loading conditions. By analysing the stress distribution around MIs and bone, clinicians can optimise implant configuration and positioning, thereby reducing the risk of implant failure [9].

While previous research has investigated general stress patterns around MIs and IZCs [6,10], specific evaluations of stress distribution for IZC under mechanical vibration have not been conducted. This study employs FEA to analyse the stress distribution patterns in bone surrounding IZC MIs under mechanical vibration. By simulating in-vitro conditions, the research aims to identify critical factors influencing implant stability and to provide insights into

optimising clinical outcomes. The findings are expected to enhance the understanding of IZC screw biomechanics, particularly under vibratory forces, and guide the development of more resilient anchorage systems for orthodontic applications.

The aim of this study was to investigate the effects of mechanical vibration on IZC MIs and the surrounding maxillary bone using FEA. The objectives were to evaluate stress distribution and displacement in cortical and trabecular bone and teeth under static and vibratory loads, and to assess the biomechanical implications of vibration-assisted force application for accelerating orthodontic tooth movement.

MATERIALS AND METHODS

This in-vitro experimental study employed FEM modelling to evaluate stress distribution and displacement around IZC MIs under static and vibratory loading conditions. The study was conducted at the Department of Orthodontics of Saveetha Dental College, Chennai, between January 2024 and March 2024. Ethical approval was not applicable as the research was simulation-based and did not involve human or animal subjects directly. Therefore, informed consent and IEC clearance were not required.

Since the study was computational, no biological subjects were involved, and hence conventional sample size calculation did not apply. However, the FEM model constructed contained 717125 nodes and 2855901 elements, ensuring adequate granularity for simulation accuracy [Table/Fig-1]. Similar anatomical FEMs with comparable mesh density have been constructed in the literature: e.g., 1.9 M elements and 2.2 M nodes in an infrazygomatic miniscrew model [11] dental-periodontium models employing ~1 M nodes and 5–6 M elements with convergence testing ; and jaw models with ~222 k nodes and ~1.5 M elements [12] confirming that our mesh of 2.85 M elements/717 k nodes provides similarly robust granularity.

Node and element details for model setup		
Name	No. of nodes	No. of elements
Maxilla bone	335613	1584478
Teeth	239084	1102363
PDL	33702	66384
Bracket and wire	101476	74706
Screw	7250	27970
Total	717125	2855901

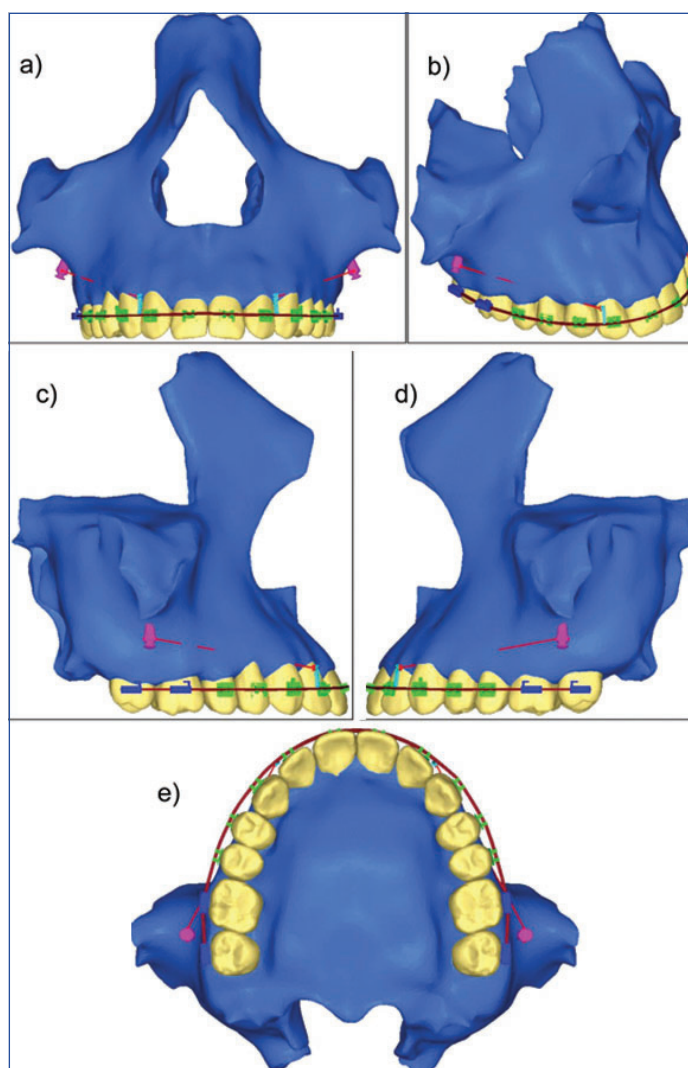
[Table/Fig-1]: Node and element for model set-up.

The inclusion criterion for model development was the representation of an anatomically accurate adult maxillary segment, including the cortical and trabecular bone, maxillary teeth, orthodontic brackets, archwire, and an IZC screw placed between the first and second molars. The model was excluded if it lacked structural integrity or geometric realism for FEM analysis.

The 3D geometry was modeled using SolidWorks 2021, followed by meshing in HyperMesh 14, and finite element simulation in Abaqus 6.14. Material properties were defined as isotropic, linearly elastic, and homogeneous, and assigned based on literature-derived values of Young's modulus and Poisson's ratio [13,14]. The boundary conditions fixed the base of the maxilla in all degrees of freedom. A static load of 250 grams was applied at the IZC implant head [1, 15], followed by vibratory force simulation ranging from 30 to 120 Hz to replicate clinical vibratory device frequencies [16-19].

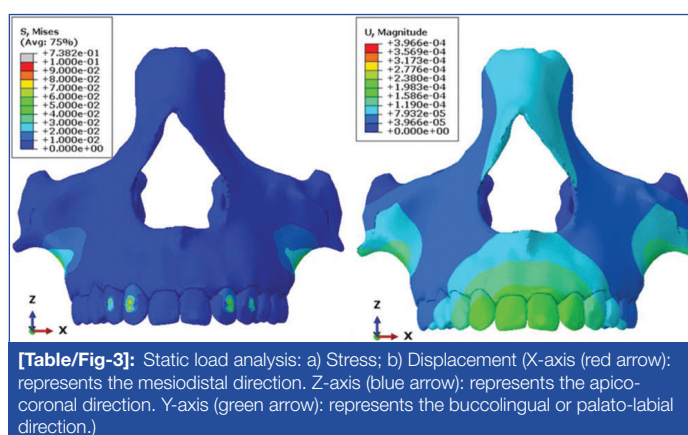
CAD model details are presented in [Table/Fig-2]. Three types of analysis were performed: (1) Static load analysis to determine baseline stress and displacement [Table/Fig-3]; (2) Modal frequency analysis to extract the first 10 natural frequencies and assess resonance risk; and (3) frequency response analysis to compare stress-displacement outcomes under vibratory loading. All models

were subjected to identical contact definitions and constraints [Table/Fig-4,5].



[Table/Fig-2]: CAD model details: a) Frontal view; b) ISO view; c) Right-side view; d) Left-side view; e) Occlusal view.

Descriptive data analysis was used to compare maximum stress (MPa) and displacement (nm) across static and vibratory conditions. No inferential statistical tests were applied due to the deterministic nature of FEM. However, findings were interpreted using engineering thresholds for material deformation and implant stability.



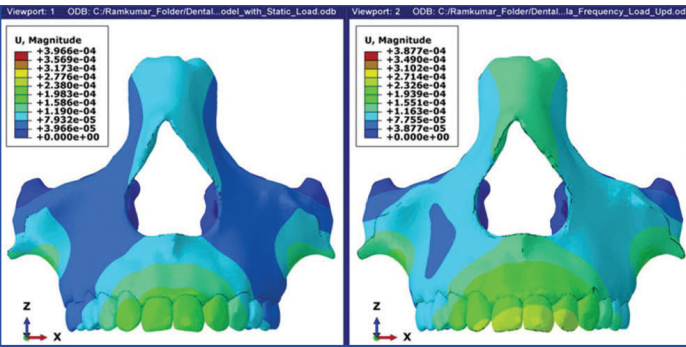
[Table/Fig-3]: Static load analysis: a) Stress; b) Displacement (X-axis (red arrow): represents the mesiodistal direction. Z-axis (blue arrow): represents the apico-coronal direction. Y-axis (green arrow): represents the buccolingual or palato-labial direction.)

RESULTS

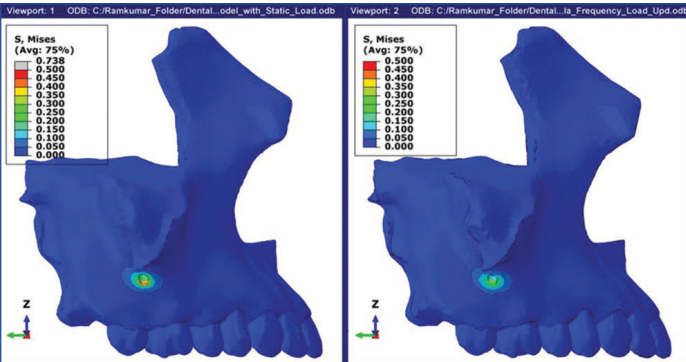
Displacement Comparison

Stress comparison

The FEM analysis was conducted to evaluate the comparative stress and displacement responses of the maxillary bone and teeth under static and vibratory loads. Initially, a static load analysis was



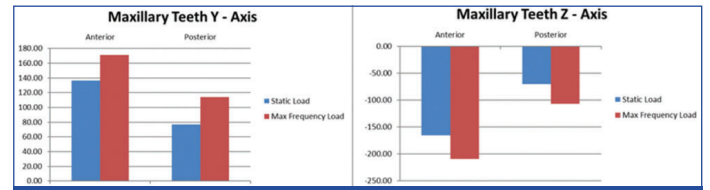
[Table/Fig-4]: Frequency response evaluation- Displacement Comparison: a) Static; b) Maximum frequency response (X-axis (red arrow): represents the mesiodistal direction. Z-axis (blue arrow): represents the apico-coronal direction. Y-axis (green arrow): represents the buccolingual or palato-labial direction.)



[Table/Fig-5]: Frequency response evaluation- stress comparison: a) Static; b) Maximum frequency response (X-axis (red arrow): represents the mesiodistal direction. Z-axis (blue arrow): represents the apico-coronal direction. Y-axis (green arrow): represents the buccolingual or palato-labial direction.)

occurs when the applied frequency matches the natural frequency, the results conclusively indicated no risk of resonance within the given frequency range. The absence of resonance ensures the structural integrity of the maxillary bone and teeth under vibratory loads.

Further, a frequency response analysis was performed to evaluate the stress and displacement responses under vibratory conditions. The results revealed minor changes in displacement and stress when compared to the static load conditions. The maximum stress in the maxillary bone reduced slightly from 0.74 MPa to 0.43 MPa, while in the teeth, stress values varied between 0.04 MPa to 0.1281 MPa, depending on the location [Table/Fig-6]. Displacement increased under vibratory conditions, particularly in the teeth. In the Y-axis direction, the anterior and posterior teeth exhibited increased displacements of 171.3 nm and 114.2 nm, respectively, compared to static conditions [Table/Fig-9]. Similarly, the Z-axis displacement increased to -210 nm and -107.1 nm for anterior and posterior teeth, respectively [Table/Fig-10,11].



[Table/Fig-9]: Teeth - displacement result summary.

Loading condition	Anterior teeth	Posterior teeth
Static load	136.40	77.07
Max frequency load	171.30	114.20

[Table/Fig-10]: Displacement in Y-axis (nanometers) of the maxillary teeth under static and vibratory loading.

Loading condition	Anterior teeth	Posterior teeth
Static load	-165.50	-70.21
Max frequency load	-210.00	-107.10

[Table/Fig-11]: Displacement in the Z-axis (nanometers) of maxillary teeth under static and vibratory loading; X-axis shows no displacement in that direction.

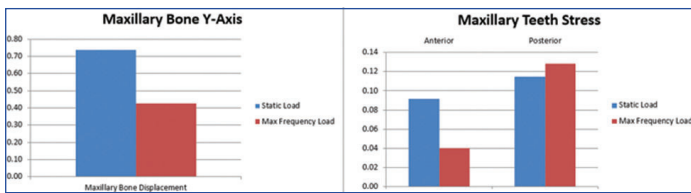
In summary, the results demonstrate that the maxillary bone and teeth exhibit structural stability under both static and vibratory load conditions. The natural frequency analysis confirmed that the system's frequencies are much higher than the applied vibratory range, eliminating the possibility of resonance. Furthermore, the frequency response analysis highlighted minimal stress changes and slight improvements in displacement under vibratory loads, suggesting that vibratory forces may positively influence tooth movement within safe limits. These findings provide valuable insights for clinical applications, particularly in dental implant stability and orthodontic treatments. However, patient-specific variations in bone stiffness could affect the results, and further investigations with real-world stress data are recommended.

DISCUSSION

Implant stability can be compromised due to factors such as excessive loading, inflammation, cortical bone thickness, proximity to tooth roots, and the design of the screw. To better understand implant failure, it is essential to study failure rates and the associated risk factors.

The present study utilised FEM analysis to evaluate the mechanical behaviour of the maxillary bone and teeth under static and vibratory loads. The findings revealed that the applied static load of 250 grams induced minimal stress and displacement, suggesting that the structural integrity of the maxilla and teeth remains unaffected by static conditions. These results align with previous studies demonstrating that low static forces applied to dental implants

performed by applying a load of 250 grams at the implant head with fixed boundary conditions. The results showed minimal stress and displacement within the maxillary bone and teeth. The maximum stress observed in the maxillary bone was 0.74 MPa, while the displacement was negligible [Table/Fig-6,7]. Similarly, for the teeth, the stress (0.09 - 0.11 MPa) and displacement values remained low under static load conditions [Table/Fig-8]. This indicated that the applied static load had no significant impact on the system, suggesting structural stability under static conditions.



[Table/Fig-6]: Maxillary bone and teeth – stress result summary.

Loading condition	Stress (MPa)
Static load	0.74
Max frequency load	0.43

[Table/Fig-7]: Stress in maxillary bone under static and vibratory loading.

Loading condition	Anterior teeth	Posterior teeth
Static load	0.09	0.1145
Max frequency load	0.04	0.1281

[Table/Fig-8]: Stress distribution (MPa) in maxillary teeth under static and vibratory loading.

To investigate the system's behaviour under vibratory loads, a modal frequency analysis was performed. The analysis aimed to identify the natural frequencies of the system and assess the risk of resonance under vibratory conditions. The first 10 natural frequencies of the system were extracted, with the lowest natural frequency observed at 1048 Hz. This value was significantly higher than the applied vibratory frequency range of 30 Hz to 120 Hz. Since resonance

and surrounding structures result in negligible stress propagation and deformation within cortical and cancellous bone regions [20,21].

The modal frequency analysis performed in this study revealed that the natural frequency of the system begins at 1048 Hz, which is significantly higher than the applied vibratory frequency range of 30-120 Hz. This observation is critical as it eliminates the risk of resonance, a phenomenon where matching external and natural frequencies could amplify displacements and induce structural failure. Resonance avoidance is particularly important in dental implant systems, where vibratory forces are introduced during mastication or orthodontic treatments [22,23]. Prior to loading the implant, it is crucial to measure the primary stability. Resonance Frequency Analysis (RFA) is one of the most widely used methods for measuring implant stability because of its dependability and non-invasiveness [23]. Similar findings have been reported in literature, where natural frequencies of implant-bone systems were found to exceed typical external load frequencies, ensuring system stability under functional loading conditions [5].

The frequency response analysis further highlighted that vibratory forces led to slight improvements in tooth displacement without significant changes in stress. Specifically, the displacement of the maxillary teeth increased in both the Y-axis and Z-axis directions compared to static loading conditions. These observations suggest that controlled vibratory loads could facilitate tooth movement, which is a principle often exploited in orthodontic therapies to accelerate tooth displacement and remodeling of periodontal structures [24]. The observed minimal stress changes under vibratory conditions are consistent with findings from earlier studies, where controlled vibratory forces improved tissue remodeling and reduced stress concentrations in the alveolar bone without compromising bone integrity [25,26].

This study demonstrates that vibratory forces applied within safe frequency ranges can enhance tooth displacement without inducing excessive stress in the maxillary bone and teeth. The absence of resonance ensures the system's stability, making vibratory techniques a promising adjunct in orthodontic and implant treatments. Future research should focus on validating these numerical findings through in vivo studies and incorporating patient-specific models to optimise clinical outcomes.

Limitation(s)

The results also underscore the importance of patient-specific parameters, such as bone stiffness and density, which could influence the natural frequency and response to vibratory forces. Variability in these factors can alter the resonance behaviour and mechanical responses, emphasising the need for personalised treatment planning in clinical settings. Furthermore, the findings encourage further investigations using experimental validation and real-world stress data to refine FEM models for more accurate predictions.

CONCLUSION(S)

The findings of this FEA study indicate that the natural frequency of the maxillary assembly, including the IZC MI, is significantly higher than the applied vibratory frequency range (30-120 Hz), effectively ruling out the risk of resonance. Comparative analysis between static and vibratory loading conditions revealed no significant difference in stress distribution within the bone and surrounding structures, suggesting that the application of vibratory forces does not compromise implant or bone integrity. However, a noticeable improvement in tooth displacement was observed under vibratory loading, indicating a potential biomechanical advantage of using mechanical vibration to enhance orthodontic tooth movement within safe limits.

Acknowledgement

We sincerely thank engineer Mr Ramkumar. M for helping in the finite element model set-up and analysis.

REFERENCES

- [1] Sreenivasagan S, Subramanian AK, Chae JM. Comparison of treatment effects during en-masse retraction of upper anterior teeth placed using mini-implants placed at infrazygomatic crest and interradicular sites: A randomized controlled trial. *Orthod Craniofac Res.* 2024;27(1):33-43.
- [2] Schwertner A, de Almeida-Pedrin RR, Poleti TMFF, Oltramari P, de Castro Conti ACF, Cotrim-Ferreira FA, et al. Biomechanical analysis of total arch maxillary distalization using infrazygomatic crest miniscrews: A finite element analysis study. *Prog Orthod.* 2024;25(1):10.
- [3] Yamaguchi M, Inami T, Ito K, Kasai K, Tanimoto Y. Mini-implants in the anchorage armamentarium: New paradigms in the orthodontics. *Int J Biomater.* 2012;2012:394121.
- [4] Song Q, Jiang F, Zhou M, Li T, Zhang S, Liu L, et al. Optimal sites and angles for the insertion of orthodontic mini-implants at infrazygomatic crest: A cone beam computed tomography (CBCT)-based study. *Am J Transl Res.* 2022;14(12):8893-902.
- [5] Pasaoglu Bozkurt A. Effects of mechanical vibration on miniscrew implants and bone: Fem analysis. *Int Orthod.* 2019;17(1):38-44.
- [6] Kovuru V, Aileni KR, Mallepally JP, Kumar KS, Sursala S, Pramod V. Factorial analysis of variables affecting bone stress adjacent to mini-implants used for molar distalization by direct anchorage-A finite element study. *J Orthod Sci.* 2023;12:18.
- [7] Pekhale N, Maheshwari A, Kumar M, Kerudi VV, Patil H, Patil B. Evaluation of stress patterns on maxillary posterior segment when intruded with mini implant anchorage: A three-dimensional finite element study. *APOS-Trends Orthod.* 2016;6(1):18-23.
- [8] Marya A. Finite element analysis and its role in orthodontics. *Adv Dent Oral Health [Internet].* 2016;2(2). Available from: <https://juniperpublishers.com/adoh/ADOH.MS.ID.555585.php>.
- [9] Ciccì M, Bramanti E, Cecchetti F, Scappaticci L, Guglielmino E, Risitano G. FEM and Von Mises Analyses of different dental implant shapes for masticatory loading distribution. *Oral Implantsol (Rome).* 2014;7(1):01-10.
- [10] 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(2):182-89.
- [11] Moga RA, Olteanu CD, Daniel BM, Buru SM. Finite elements analysis of tooth-a comparative analysis of multiple failure criteria. *Int J Environ Res Public Health [Internet].* 2023;20(5). Available from: <http://dx.doi.org/10.3390/ijerph20054133>.
- [12] Tang R, Cui Q, Ren S, Unurjargal K, Li J. Study on finite element modeling method of oral soft and hard tissues based on CBCT images and digital scanning technology. 2024 Jun 28 [cited 2025 Jul 11]; Available from: <https://www.researchsquare.com/article/rs-4434940/latest.pdf>.
- [13] Harsha SB, Riyaz K, Manjappa LS, Talwar A, Ravuru D, Narasimha Mogaveera NK, et al. Comparative analysis of displacements and stress produced by square and rectangular bracket slot with incremental torque: A finite element analysis study. *Cureus.* 2024;16(10):e71528.
- [14] Boggio A, Jamilian A, Manni A, Gastaldi G, Farjaminejad R, Hasani M, et al. The effect of a manni telescopic herbst appliance with four miniscrews (STM4) on the treatment of a class II division I malocclusion: A 3D finite element study. *Oral.* 2025;5(2):27.
- [15] Paul P, Mathur AK, Chitra P. Stress distribution patterns in mini-implant and bone in the infra-zygomatic crest region at different angulations: A finite element study. *J World Fed Orthod.* 2021;10(1):29-34.
- [16] Woodhouse NR, DiBiase AT, Johnson N, Slipper C, Grant J, Alsaleh M, et al. Supplemental vibrational force during orthodontic alignment. *J Dent Res.* 2015;94(5):682-89.
- [17] Miles P, Smith H, Weyant R, Rinchuse DJ. The effects of a vibrational appliance on tooth movement and patient discomfort: A prospective randomised clinical trial. *Aust Orthod J.* 2023;28(2):213-18.
- [18] Pavlin D, Anthony R, Raj V, Peter T. Gakunga Cyclic loading (vibration) accelerates tooth movement in orthodontic patients: A double-blind, randomized controlled trial. *Semin Orthod.* 2015;21(3):187-94.
- [19] Shipley T, Farouk K, El-Bialy T. Effect of high-frequency vibration on orthodontic tooth movement and bone density. *J Orthod Sci.* 2019;8(1):15.
- [20] nokar S, Jalali H, Nozari F, Arshad M. finite element analysis of stress in bone and abutment-implant interface under static and cyclic loadings. *Front Dent.* 2020;17(21):01-08.
- [21] Tezerigener HA, Özalp Ö, Altay MA, Sindel A. Comparison of stress distribution around all-on-four implants of different angulations and zygoma implants: A 7-model finite element analysis. *BMC Oral Health.* 2024;24(1):176.
- [22] Hosseini-Faradonbeh SA, Katoozian HR. Biomechanical evaluations of the long-term stability of dental implant using finite element modeling method: A systematic review. *J Adv Prosthodont.* 2022;14(3):182-202.
- [23] Quesada-García MP, Prados-Sánchez E, Olmedo-Gaya MV, Muñoz-Soto E, González-Rodríguez MP, Vallecillo-Capilla M. Measurement of dental implant stability by resonance frequency analysis: A review of the literature. *Med Oral Patol Oral Cir Bucal.* 2009;14(10):e538-e546.
- [24] Kalajzic Z, Peluso EB, Utraja A, Dymont N, Nihara J, Xu M, et al. Effect of cyclical forces on the periodontal ligament and alveolar bone remodeling during orthodontic tooth movement. *Angle Orthod.* 2014;84(2):297-303.

- [25] Leethanakul C, Suamphan S, Jitpukdeebodintra S, Thongudomporn U, Charoemratrote C. Vibratory stimulation increases interleukin-1 beta secretion during orthodontic tooth movement. *Angle Orthod.* 2016;86(1):74-80.
- [26] Vibratory stimulus and accelerated tooth movement: A critical appraisal. *J World Fed Orthod.* 2018;7(3):106-12.

PARTICULARS OF CONTRIBUTORS:

1. Resident, Department of Orthodontics, Saveetha Dental College and Hospitals, Saveetha Institute of Medical and Technical Sciences, Saveetha University, Chennai, India.
2. Professor and Head, Department of Orthodontics, Saveetha Dental College and Hospitals, Saveetha Institute of Medical and Technical Sciences, Saveetha University, Chennai, India.
3. Assistant Professor, Department of Orthodontics, Saveetha Dental College and Hospitals, Saveetha Institute of Medical and Technical Sciences, Saveetha University, Chennai, India.

NAME, ADDRESS, E-MAIL ID OF THE CORRESPONDING AUTHOR:

Dr. Aravind Kumar Subramanian,
162, Poonamallee High Road, Chennai-600077, Tamil Nadu, India.
E-mail: aravindkumar@saveetha.com

PLAGIARISM CHECKING METHODS: [\[Jain H et al.\]](#)

- Plagiarism X-checker: Feb 27, 2025
- Manual Googling: Sep 02, 2025
- iThenticate Software: Sep 04, 2025 (6%)

ETYMOLOGY: Author Origin**EMENDATIONS:** 6**AUTHOR DECLARATION:**

- Financial or Other Competing Interests: None
- Was informed consent obtained from the subjects involved in the study? Yes
- For any images presented appropriate consent has been obtained from the subjects. Yes

Date of Submission: **Feb 26, 2025**Date of Peer Review: **May 13, 2025**Date of Acceptance: **Sep 06, 2025**Date of Publishing: **Jun 01, 2026**