Journal of Orthodontic and Periodontal Biomaterials Research

2024, Volume 4, Issue 1, Page No: 79-88
Copyright CC BY-NC-SA 4.0
Available online at: https://galaxypub.co/

Galaxy Publication

Effect of Force Optimization on the Longevity and Stability of Orthodontic Mini-Implants

Arif F. Wijaya[1], Minh Tran[1], Anh Hoang[1], Rizky Rahman[2], Minh R. Ismail[1]*, Anh V. Nguyen[1]

[1]Department of Periodontology, Faculty of Dentistry, Vietnam National University, Hanoi, Thailand.
[2]Department of Orthodontics and Dentofacial Orthopedics, School of Dentistry, University of the Philippines, Manila, Thailand.

***E-mail** ✉ minh.ismail@gmail.com

Received: 19 October 2024; **Revised:** 17 January 2024; **Accepted:** 18 January 2024

ABSTRACT

This research explores how titanium (Ti6Al4V) mini-implants (MIs) respond mechanically to different orthodontic forces using finite element analysis (FEA), with the goal of assessing their durability and functional performance under conditions mimicking clinical use. Applying forces within an optimal range is crucial for ensuring implant stability and protecting surrounding bone tissue. A standard MI (12 mm length, 2 mm diameter) was simulated in FEA. The mandible was reconstructed in three dimensions from CT scans using SpaceClaim 2023.1 and meshed with 10-node tetrahedral elements in ANSYS Workbench. Material characteristics were taken from the literature, and the implant–bone interface was modeled with nonlinear frictional contact. Forces of 2 N and 10 N were applied at a 30° angle to replicate typical clinical loading. The analysis focused on implant displacement, stress distribution (von Mises), strain levels, fatigue life, and safety margins. When loaded with 2 N, the implant showed minimal movement (0.0328 mm) and could withstand approximately 445,000 cycles with a safety factor of 4.84. Increasing the force to 10 N drastically reduced fatigue life to 1,546 cycles, while stress and strain concentrations rose sharply (6.468×10^5 MPa), indicating a higher probability of implant failure and bone damage. These results identify the threshold at which excessive force jeopardizes MI stability and peri-implant bone integrity. The findings emphasize that maintaining orthodontic forces near 2 N is critical for prolonging MI lifespan and safeguarding bone. Forces beyond this optimal range, such as 10 N, significantly compromise implant durability and increase failure risk, highlighting the importance of careful force management in orthodontic treatments. This study offers practical insights to improve MI performance and clinical outcomes.

Keywords: Stress distribution, Cortical bone, Fracture, Finite element method (FEM), Orthodontic forces, Mini-implant

How to Cite This Article: Wijaya AF, Tran M, Hoang A, Rahman R, Ismail MR, Nguyen AV. Effect of Force Optimization on the Longevity and Stability of Orthodontic Mini-Implants. J Orthod Periodontal Biomater Res. 2024;4(1):79-88. https://doi.org/10.51847/uB0vQ5YG8T

Introduction

Orthodontic mini-implants (MIs) have become widely adopted as temporary anchorage devices (TADs) because they provide dependable support for various tooth movements. However, their clinical reliability is still limited by failure rates reported between 5% and 30%, which are influenced by insertion site, individual patient factors, and the magnitude or direction of applied forces [1–4]. Most failures are linked to inadequate primary stability, a critical factor that ensures the implant maintains firm mechanical engagement with surrounding bone under orthodontic loading [5, 6]. Cortical bone thickness at the implantation site plays a major role in this stability, with studies suggesting higher success rates in thicker bone areas such as the anterior palate compared to buccal or interradicular regions [7]. Yet, the effect of varying cortical bone thickness on MI performance, particularly under different force levels, remains poorly understood. Insufficient cortical support can generate localized stress concentrations, trigger microfractures, and ultimately cause implant loosening, undermining treatment effectiveness [8, 9].

The distribution of mechanical stress during orthodontic treatment is heavily influenced by cortical bone characteristics. While thin cortical layers are known to experience higher stress under loading, the consequences for MI stability and bone adaptation have not been fully clarified [8]. Optimizing implant design, insertion angle, and site selection is therefore essential to achieve reliable mechanical stability [10–12]. Recent technological advances—including finite element analysis (FEA), 3D digital simulations, and innovative material designs—allow clinicians and researchers to better predict stress patterns and customize treatment strategies [13–18]. For example, studies have demonstrated that inserting implants at an oblique angle can lower stress concentrations in cortical bone, potentially reducing failure risk [4, 19]. Moreover, the use of titanium alloys with mechanical properties closer to natural bone minimizes stress shielding, further supporting long-term implant stability [20, 21].

Despite these advancements, few studies have systematically investigated how cortical bone thickness interacts with applied orthodontic forces to affect MI stability. This study addresses that gap by analyzing stress distribution and mechanical behavior of MIs across varying cortical thicknesses using FEA. The findings aim to guide clinicians in selecting optimal implant sites and force levels to enhance both the durability of MIs and the health of surrounding bone.

Materials and Methods

Geometric modeling

A commercially available titanium (Ti6Al4V) MI (Jeil Medical Corporation, Seoul, South Korea), measuring 12 mm in length and 2.0 mm in diameter, was digitally modeled for finite element analysis (FEM) **(Figure 1a)**. High-resolution computed tomography (CT) scans of the mandible (DEXIS, Biberach, Germany) were obtained and processed to capture accurate anatomical geometry **(Figure 1b)**. Using SpaceClaim 2023.1, three-dimensional solid models of the mandible and MI were reconstructed and assembled into a comprehensive FEM model **(Figure 1c)**, allowing precise simulation of implant–bone interactions under orthodontic loading.

Figure 1. Overview of the finite element modeling process for an orthodontic mini-implant (MI) and surrounding bone structures: (a) A commercially available titanium MI with a 2 mm diameter was digitally modeled using the finite element method; (b) Mandibular geometry obtained from CT scans and converted to STL format; (c) The assembled 3D model discretized into finite elements (left) with a detailed view of the MI region and adjacent anchorage area (right).

Simulation setup

The MI was positioned in the interproximal region between the premolar and molar. The complete model was imported into ANSYS 2021 (ANSYS, Inc., Canonsburg, PA, USA) for finite element analysis and discretized using automatic 10-node tetrahedral elements. All components, including bone, teeth, the periodontal ligament, and the MI, were treated as linear elastic, isotropic, and homogeneous materials. Material properties were assigned based on previously published data **(Table 1)**. The tooth–periodontal ligament interface was assumed to be perfectly bonded, while the MI–bone interface was modeled with frictionless contact (friction coefficient = 0).

Table 1. The material properties utilized in the finite element analysis were modeled based on specific characteristics to ensure accuracy and realism [22]

Material/Component	Elastic Modulus (MPa)	Poisson's Ratio
Bracket	380,000	0.19
Mini-Implant	110,000 200,000	0.3
Tooth	84,100	0.2
PDL	68.9	0.45
Cortical bone	17,000	0.3
Spongy bone	350	0.25

Orthodontic loading was simulated by applying forces of 2 N and 10 N at a 30° inclination from the vertical (Y) axis. These forces were transmitted from the MI toward the molar through the connector tube, closely replicating the biomechanical conditions of molar intrusion supported by skeletal anchorage. This oblique force application mirrors clinical load distribution during orthodontic procedures and enables a realistic evaluation of stress and strain patterns within the periodontal ligament, alveolar bone, and adjacent tissues.

FEA model specifications

Table 2 summarizes the key parameters used in the finite element analysis (FEA) for this study, providing a detailed overview of the modeling setup. Each parameter listed in the table corresponds to specific aspects of the FEA model, as briefly described below:

Table 2. Finite element analysis model parameters

FEA Parameter	Property
Property discretization (nodes/elements)	356,422/229,672
Element	10 node tetrahedron
Software	ANSYS Workbench 19.2, Canonsburg, PA, USA
Material model	Isotropic, homogeneous, linear
Contact model	Friction (nonlinear, friction coefficient = 0 between mini-implant and bones), bonded (linear)
Loading	Oblique (0.1–10 N) (30°, 45°, 60°)
Mini-implant type	Threaded, diameter: 2.0 mm, 12.0 mm length
Boundary conditions	100% osseointegration

1. Discretization (nodes/elements): This parameter reflects the total number of nodes and elements incorporated in the finite element model. A higher density of nodes and elements allows for a more precise representation of the geometry and the interactions within the model.
2. Element type: This refers to the specific finite elements employed, which in this study are 10-node tetrahedral elements. These elements subdivide the geometry and enable calculation of the structural response under applied loads.
3. Software: The computational platform used to run the simulations is ANSYS.
4. Material properties: The model assumes isotropic, homogeneous, and linear material behavior, meaning the material characteristics remain consistent in all directions and positions, and deformation responds linearly to applied loads.
5. Contact modeling: Interactions between different components are simulated using nonlinear frictional contact for the MI–bone interface, while other contacts, such as between teeth and periodontal ligament, are considered perfectly bonded.
6. Loading conditions: Orthodontic forces were applied obliquely, with magnitudes of 2 N and 10 N, replicating real-world clinical scenarios.
7. MI specifications: The characteristics of the mini-implants, including dimensions and thread design, are defined here.
8. Boundary conditions: Full osseointegration is assumed, indicating that the MI is completely anchored to the bone, with no relative motion at the interface.

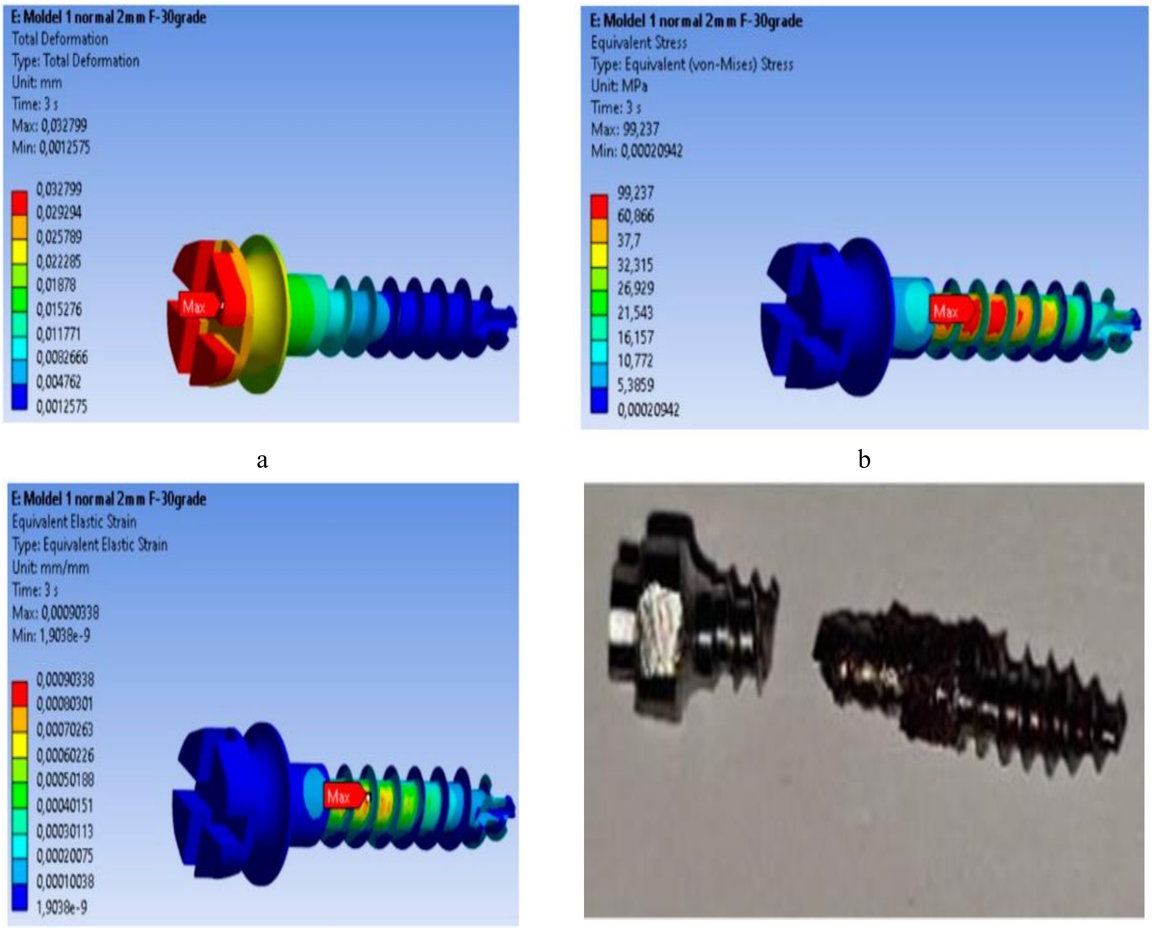
Results and Discussion

A structural static analysis was conducted under the assumption of an undamped system and uniform material properties. Stiffness was modeled using the linear, isotropic material characteristics. Results for the 2 N loading condition are described below. The MI was inserted 7 mm into the bone, positioned perpendicular to the cortical and cancellous surfaces (90°), and subjected to a 30° inclined orthodontic force relative to the vertical axis (Y).

Key findings: Displacement, von mises stress, and strain

Finite element analysis (FEA) revealed detailed insights into the MI's mechanical response under applied orthodontic loads, focusing on total displacement, von Mises stress, and equivalent strain. The displacement results **(Figure 2a)** indicated a maximum movement of 0.032799 mm at the free end of the MI, consistent with the expectation that bending forces produce the largest deflection at unconstrained regions, while the threaded portion remains anchored. Von Mises stress distribution **(Figure 2b)** showed a peak stress of 99.237 MPa localized in the threaded section, aligning with common failure zones observed clinically. Equivalent strain analysis **(Figure 2c)** revealed a maximum strain of 0.00090338, also concentrated in the threaded area. The concurrence of high stress and strain in this region underscores its vulnerability to fatigue and potential failure under repeated loading.

The practical relevance of these simulations is confirmed in **Figure 2d**, which displays a fractured MI occurring precisely at the location predicted by the FEA to experience the highest stress. This agreement between computational predictions and physical failure validates the model and confirms the accuracy of the stress and strain distributions obtained from the analysis.



a b

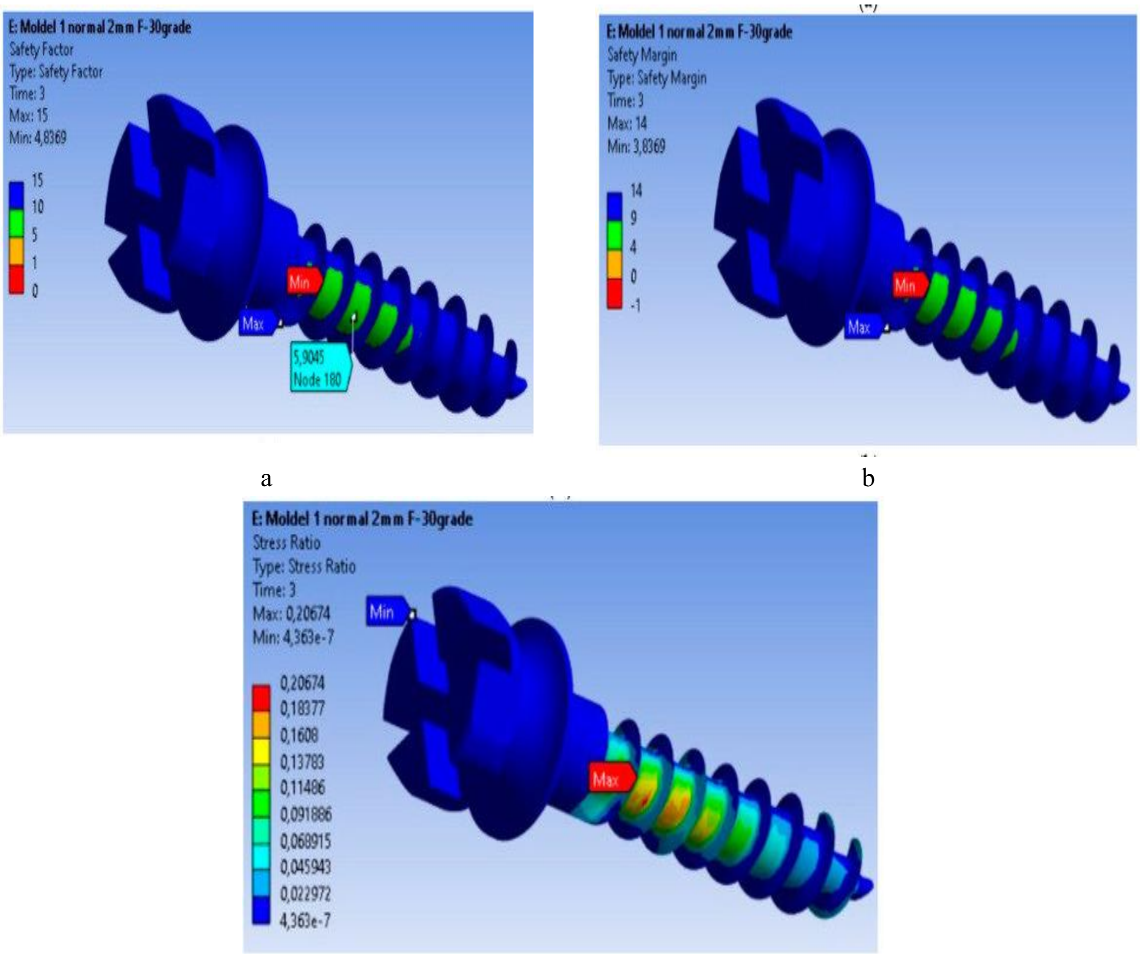
c d

Figure 2. Mechanical response of the mini-implant under a 2 N load applied at a 30° angle: (a) Overall displacement; (b) von Mises stress distribution; (c) Equivalent strain mapping; (d) Physical fracture of the MI upon removal, occurring precisely at the location predicted by the FEA as the highest stress concentration.

Safety assessment of the MI: Safety factors, margins, and stress ratios

The calculated safety factors for the MI are illustrated in **Figure 3a**. The lowest safety factor, 4.8369, occurs at the node experiencing the highest stress, indicating that the implant safely meets structural strength requirements under a 2 N orthodontic load. This factor represents the ratio of the material's yield strength to the von Mises stress at the critical point. For this study, the yield strength of the titanium MI was experimentally determined to be 480 MPa.

The corresponding safety margin is presented in **Figure 3b**, with a minimum value of 3.8369, further confirming that the MI remains well within safe operational limits. **Figure 3c** depicts the stress ratio, defined as the calculated stress divided by the material's yield strength. The maximum stress ratio is 0.20674, located at the node with the peak von Mises stress, highlighting that the MI operates far below its material limits under the applied 2 N loading condition.



a

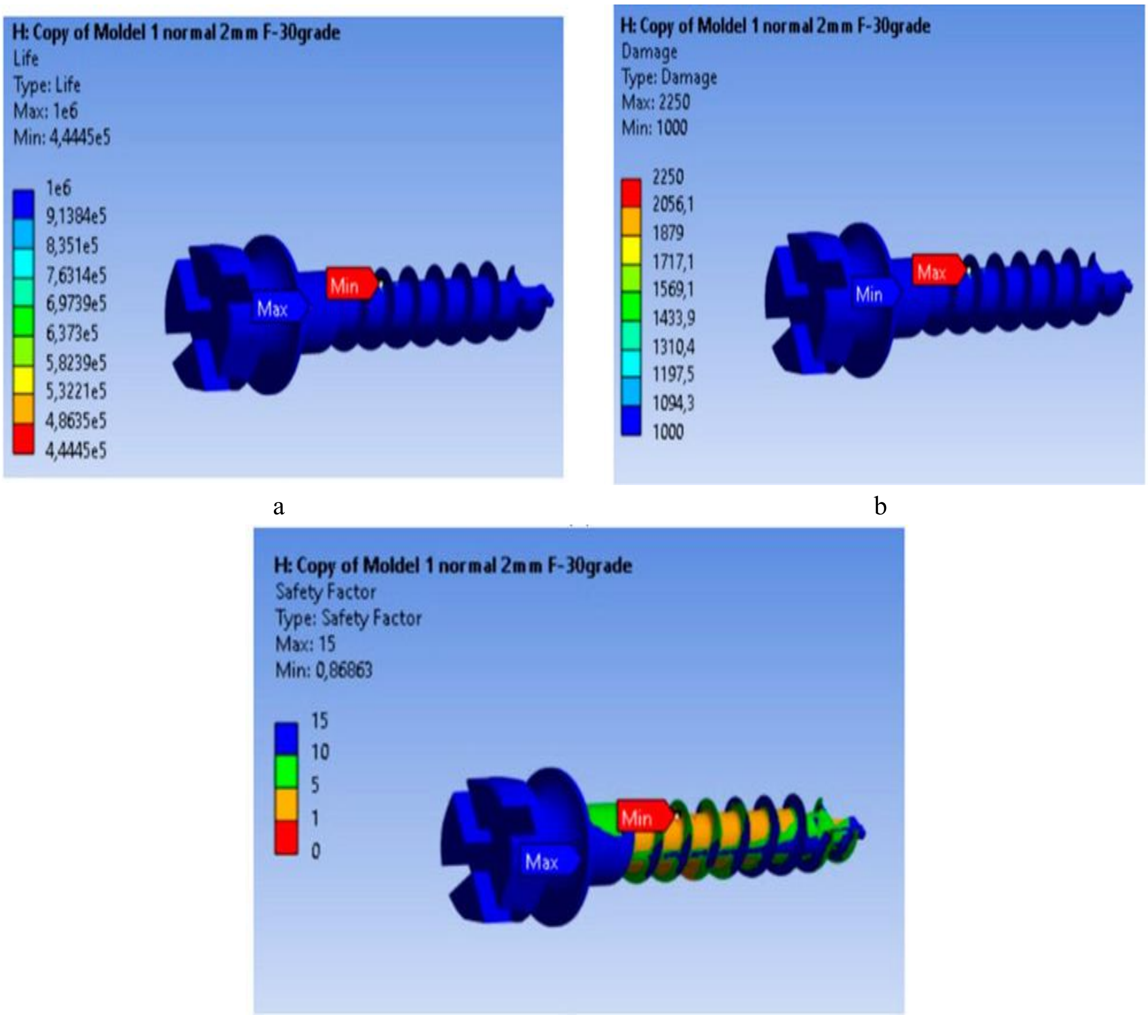
b

c

Figure 3. Evaluation of mini-implant safety under a 2 N load: (a) Safety factor distribution (σc/σMax); (b) Corresponding safety margin ((σc/σMax) − 1); (c) Stress ratio (σMax/σc)

Fatigue performance of the mini-implant at 2 N: Predicted lifespan and failure risk

Fatigue behavior of the MI was analyzed under a symmetric alternating loading pattern, using the ASME elliptical approach [23]. Lifespan predictions were based on von Mises equivalent stress, applying a scale factor of 1. The analysis indicates that the implant can endure approximately 444,500 cycles without experiencing fatigue damage under a 2 N orthodontic load. Examination of Figures 4a–c shows that the lowest predicted fatigue life is concentrated in a very small region, while most of the implant material demonstrates a considerably longer fatigue life of about 1,000,000 cycles. The highest local factor, 2,250, corresponds to the node where fatigue life is minimal, representing roughly 2.5 times the surrounding material's value. Fatigue safety factors, illustrated in **Figure 4c**, show a minimum value of 0.86863 at the most stressed node, highlighting the region most prone to fatigue-related failure.



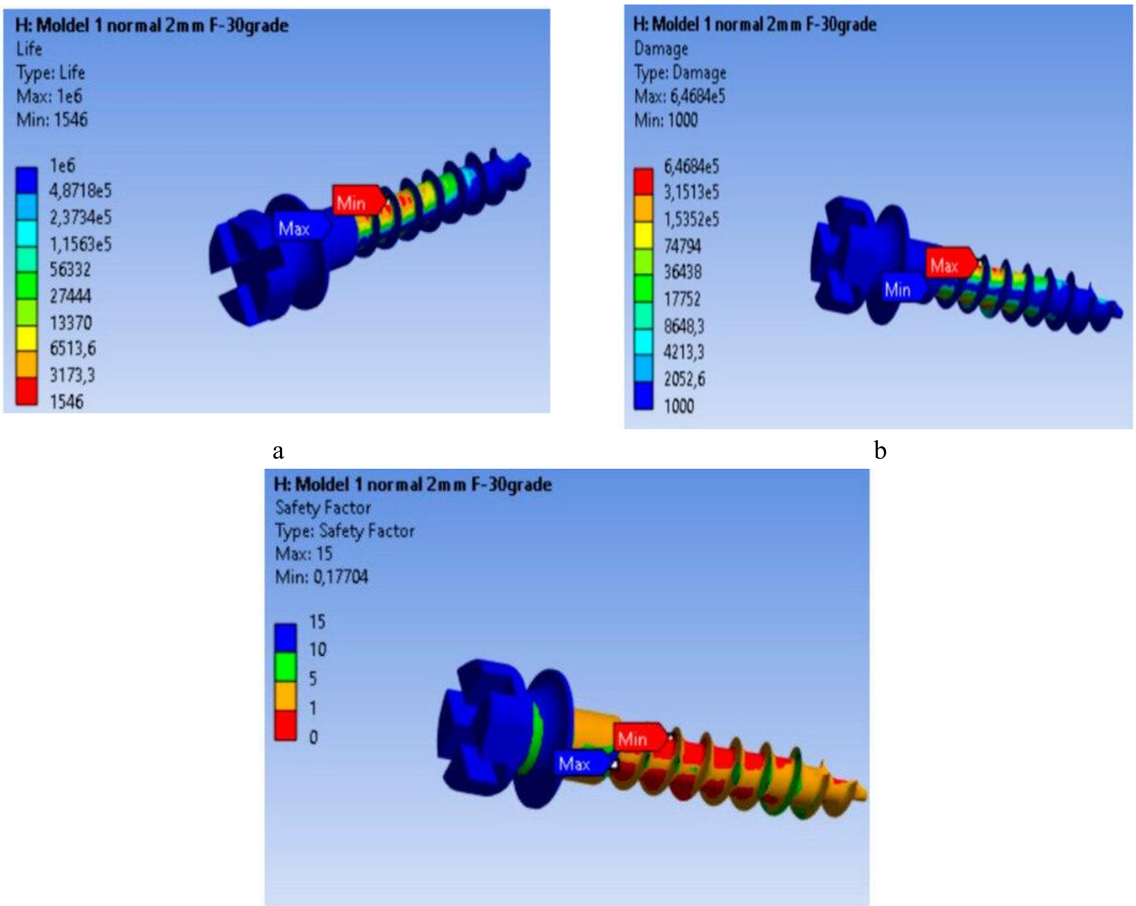
a b

c

Figure 4. Outcomes for mini-implant endurance under a 2 N force: (a) Endurance; (b) Failure; (c) Fatigue safety factors

Fatigue behavior of the MI under a 10 N orthodontic load: Life span, safety margins, and failure assessment

The operating duration of the mini-implant under safe fatigue conditions for a 10 N orthodontic force is illustrated in **Figure 5a**. The results indicate that the MI can function without failure for 1,546 cycles, which is 287.48 times shorter than under a 2 N load. Therefore, a 10 N force significantly reduces the defect-free lifespan of the MI compared to a 2 N load. The failure pattern is depicted in **Figure 5b**, where the maximum value of 6.468×10^5 at the node with the lowest fatigue life is 6.468×10^2 times higher than in the rest of the implant. The corresponding fatigue safety factors are shown in **Figure 5c**, with the lowest safety factor of 0.17704 occurring at the node experiencing the highest stress.



a b

c

Figure 5. Mini-implant performance results under a 10 N force: (a) Endurance; (b) Failure; (c) Fatigue safety factors

When subjected to a 10 N load **(Table 3)**, the mini-implant exhibits a markedly reduced lifespan, a greater extent of material affected, and lower fatigue safety margins compared to the 2 N scenario, indicating a higher risk of structural failure. These findings emphasize the critical role of applying controlled forces in orthodontic procedures to maximize implant longevity.

Table 3. Comparison of 2 N and 10 N force effects on the mini-implant

Parameter	2 N Force	10 N Force
Endurance (operating cycles)	4.445×10^5 cycles	1546 cycles (287.48 times lower)
Volume of material affected	Small volume of material	Larger volume of material
Minimum fatigue safety factor	0.86863 (located at most stressed node)	0.17704 (located at most stressed node)
Maximum stress concentration	2250 MPa (2.5× higher than the rest of the material)	6.468×10^5 MPa (646.8× higher than the rest of the material)

The deformation tolerance of orthodontic mini-implants (MIs) is a key factor in maintaining their functional performance and longevity during treatment, making it essential to understand the variables that influence these limits for optimized clinical use.

Applying a 2 N force to orthodontic MIs plays a crucial role in treatment outcomes, as it directly affects the implant's stability and efficiency in facilitating tooth movement. Studies indicate that a 2 N loading force falls within the clinically acceptable range for immediate loading, which is vital for effective anchorage during orthodontic procedures [4]. Clinically, this force range is frequently applied in procedures such as canine retraction, where recommended forces generally range from 1.5 N to 2.5 N [24]. The biomechanical impact of a 2 N force on MIs has been extensively studied, and evidence supports that this level is safe for immediate loading. Finite element analyses (FEA) show that stress distribution around MIs is strongly influenced by both the magnitude of applied forces and the properties of the surrounding bone [25]. Although MIs can tolerate immediate loading, excessive forces may lead to complications, including displacement or implant failure [26].

FEA is a powerful computational tool for predicting structural behavior under various loading conditions, yet its accuracy relies heavily on the assumptions within the model. Assuming 100% osseointegration simplifies simulations but overlooks biological variability, which can result in inaccurate predictions of stress distribution, load transfer, and potential failure patterns in implants [27].

Displacement, stress, strain, and safety factor analysis in mini-implants

Displacement analysis shows a maximum total displacement of 0.032799 mm at the MI’s end, indicating a localized area of high stress concentration. This measurement helps evaluate the extent of bending and deformation experienced by the implant during orthodontic loading.

Von Mises stress analysis identified a peak stress of 99.237 MPa in the threaded region, pointing to a critical area that could represent the MI’s failure point. These findings underscore the importance of optimizing design and manufacturing techniques to enhance the implant’s resilience to such stress levels and reduce failure risk.

Regarding equivalent linear strains, the highest measured strain of 0.00090338 MPa in the threaded region provides a quantitative assessment of material deformation, which can inform improvements in MI design and material selection to enhance stability.

Safety factor interpretation shows that the minimum value meets established strength requirements, indicating the MI possesses sufficient mechanical integrity to withstand orthodontic forces, thereby minimizing the risk of deformation or fracture.

The stress ratio—defined as the ratio of applied stress to the material’s yield strength—offers insight into how close the MI operates to its structural limits. The maximum stress ratio, occurring at nodes with the highest equivalent stress, highlights critical points for evaluating the implant’s stability and overall structural safety.

The fatigue evaluation indicated that failure occurs at a specific node of the mini-implant (MI), where the minimum fatigue life was observed, highlighting a potential structural weak point that could guide future improvements in design and manufacturing. The estimated lifespan of roughly 4.445×10^5 cycles reflects the MI’s reliability and stability under repeated loading. The analysis also shows that the MI becomes considerably more vulnerable under higher loads, suggesting that excessive orthodontic forces may need to be limited to prevent structural damage. The concentration of failure at this node, combined with the reduction in fatigue safety factors, underscores a significant decrease in device reliability under increased loading conditions.

These findings stress the importance of tailoring orthodontic forces to the mechanical capabilities of each MI to prevent damage and ensure treatment safety. Nienkemper et al. highlighted that the duration of applied loading critically influences MI displacement, indicating that mechanical behavior is affected not only by the force magnitude but also by how long it is applied [26]. Studies suggest that MIs are designed to tolerate certain force thresholds, with optimal loads typically recommended around 3.75–4.5 N [25].

Future research directions

The current results provide a basis for future investigations into the mechanical performance of MIs and the development of strategies or technologies to enhance their stability and clinical effectiveness. Such research could involve additional computational simulations or experimental tests to validate and expand on these findings.

Study limitations and suggestions for future work

(i) Single implant length (12 mm): This study only assessed MIs of 12 mm in length; future studies should examine a broader range of lengths to better understand how length affects mechanical performance.

(ii) Simplified bone models: Bone was modeled as linear, elastic, and isotropic, which does not fully capture the complex, heterogeneous nature of cortical and cancellous bone.

(iii) Use of only titanium MIs (Ti6Al4V): No comparison with other materials, such as 316L stainless steel, was performed, limiting insights into how material choice might impact long-term performance and clinical outcomes.

(iv) Idealized insertion and loading conditions: The study assumed perfect MI insertion and full osseointegration (100%), whereas clinical conditions vary, and incomplete osseointegration could affect implant behavior and stability.

Conclusion

The findings of this study demonstrate that keeping orthodontic forces near 2 N is essential for maintaining the mechanical integrity and long-term function of mini-implants (MIs) in clinical practice. Under this load, finite element analysis predicts that MIs can withstand about 445,000 cycles, with stress and strain levels staying safely within the limits of both cortical and cancellous bone. Conversely, a 10 N force dramatically shortens the implant's lifespan to roughly 1,546 cycles and generates much higher stress and strain, increasing the likelihood of implant failure and potential damage to the surrounding cortical bone.

These observations highlight the importance of carefully managing applied forces during orthodontic treatment to enhance MI durability and protect peri-implant bone, allowing clinicians to optimize treatment outcomes while minimizing risks.

Acknowledgments: None

Conflict of interest: None

Financial support: None

Ethics statement: None

References

1. Malik F, Khan F, Ali S, Rana F, Haq H, Hussain M. Factors affecting success and failure of orthodontic mini-implants: A retrospective review. Prof Med J. 2023;30(2):285–91.
2. Ravi J, Duraisamy S, Rajaram K, Kannan R, Arumugam E. Survival rate and stability of surface-treated and non-surface-treated orthodontic mini-implants: A randomized clinical trial. Dent Press J Orthod. 2023;28(2):e2321345.
3. Jeong J, Kim J, Lee N, Kim Y, Lee J, Kim T. Analysis of time to failure of orthodontic mini-implants after insertion or loading. J Korean Assoc Oral Maxillofac Surg. 2015;41(4):240.
4. Popa A, Dehelean C, Călniceanu H, Watz C, Brad S, Sinescu C, et al. A custom-made orthodontic mini-implant—Effect of insertion angle and cortical bone thickness on stress distribution with a complex in vitro and in vivo biosafety profile. Materials. 2020;13(21):4789.
5. Singh J, Singh S, Gupta A. Comparative evaluation of primary stability of two different types of orthodontic mini-implants. J Pharm Bioallied Sci. 2021;13(Suppl 1):S128–35.
6. Dutta S, Bhat J. Orthodontic mini implant and their primary stability: A comparative study. Int J Appl Dent Sci. 2022;8(2):642–4.
7. Arqub SA, Gandhi V, Mehta S, Palo L, Upadhyay M, Yadav S. Survival estimates and risk factors for failure of palatal and buccal mini-implants. Angle Orthod. 2021;91(6):756–63.
8. Filipová D, Filipi V, Kamínek M. Proclination-induced changes in the labial cortical bone thickness of lower incisors. Bratisl Med J. 2019;120(3):155–60.
9. Kovuru 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(1):18.
10. Son E, Kim S, Hong C, Chan V, Sim H, Ji S, et al. A study on the morphologic change of palatal alveolar bone shape after intrusion and retraction of maxillary incisors. Sci Rep. 2020;10(1):14454.
11. Thongudomporn U, Charoemratrote C, Jearapongpakorn S. Changes of anterior maxillary alveolar bone thickness following incisor proclination and extrusion. Angle Orthod. 2014;85(4):549–54.
12. Pan C, Liu P, Tseng Y, Chou S, Wu C, Chang H. Effects of cortical bone thickness and trabecular bone density on primary stability of orthodontic mini-implants. J Dent Sci. 2019;14(4):383–8.
13. Yu W, Tsai M, Yu J, Huang H, Hsu J. Bone quality affects stability of orthodontic miniscrews. Sci Rep. 2022;12(1):2849.
14. Centeno A, Fensterseifer C, Chami V, Ferreira E, Marquezan M, Ferrazzo V. Correlation between cortical bone thickness at mini-implant insertion sites and age of patient. Dent Press J Orthod. 2022;27(2):e222098.
15. Marquezan M, Mattos C, Sant'Anna E, Souza M, Maia L. Does cortical thickness influence the primary stability of miniscrews?: A systematic review and meta-analysis. Angle Orthod. 2014;84(6):1093–103.

16. Hatia A, Doldo T, Parrini S, Chisci E, Cipriani L, Montagna L, et al. Accuracy and completeness of ChatGPT-generated information on interceptive orthodontics: A multicenter collaborative study. J Clin Med. 2024;27(3):735.
17. Sant'Anna EF, Araújo MTS, Nojima LI, Cunha ACD, Silveira BLD, Marquezan M. High-intensity laser application in orthodontics. Dent Press J Orthod. 2017;22(2):99–109.
18. Weir T. Clear aligners in orthodontic treatment. Aust Dent J. 2017;62(Suppl 1):58–62.
19. Rito-Macedo F, Barroso-Oliveira M, Paranhos LR, Rodrigues-Brum J, Pereira-Lima IF, Gomes-França FM, et al. Implant insertion angle and depth: Peri-implant bone stress analysis by the finite element method. J Clin Exp Dent. 2021;10(11):1167–73.
20. Panaite T, Savin C, Olteanu ND, Karvelas N, Romanec C, Vieriu RM, et al. Heat treatment's vital role: Elevating orthodontic mini-implants for superior performance and longevity—Pilot study. Dent J. 2024;12(3):103.
21. Cozzani M, Pasini M, Zallio F, Ritucci R, Mutinelli S, Mazzotta L, et al. Comparison of maxillary molar distalization with an implant-supported distal jet and a traditional tooth-supported distal jet appliance. Int J Dent. 2014;10:1–7.
22. Benaissa A, Merdji A, Bendjaballah MZ, Ngan P, Mukdadi OM. Stress influence on orthodontic system components under simulated treatment loadings. Comput Methods Programs Biomed. 2020;10:195–210.
23. ASME Elliptical. Available from: https://www.asme.org/ (accessed 1 June 2024).
24. Ghorab H. Finite element study to evaluate the stress around mini-implant during canine retraction using continuous and interrupted orthodontic forces. Open Access Maced J Med Sci. 2023;11(1):36–43.
25. Alrbata R, Momani M, Al-Tarawneh A, Ihyasat A. Optimal force magnitude loaded to orthodontic microimplants: A finite element analysis. Angle Orthod. 2015;86(2):221–6.
26. Nienkemper M, Handschel J, Drescher D. Systematic review of mini-implant displacement under orthodontic loading. Int J Oral Sci. 2013;6(1):1–6.
27. Klein CP, van der Waerden J. The effect of osseointegration on the mechanical stability of dental implants: A finite element analysis. J Biomech. 2015;48(1):1–7.