DOI: 10.71762/kh7j-3n24

Research Paper

Numerical Simulation of Realistic Implants and Implant Prosthesis of Molar Teeth Model by 3D Finite Element Analysis Method During Chewing Cycle

Shahin Heidari[1*], Neda Nasr[2]
[1]Bone and Joint Diseases Research Center, Department of Orthopedic Surgery, School of Medicine, Shiraz University of Medical Sciences, Shiraz, Iran
[2]Department of Periodontics, School of Dentistry, Shiraz Islamic Azad University, Shiraz, Iran
[*]Email of Corresponding Author: Heidari.shahin1@gmail.com
Received: March 2, 2024; Accepted: May 8, 2024

Abstract
In recent years, dental materials and implantology advancements have significantly improved the field. Precisely, focus has been placed on optimizing implant design, surface characteristics, and the implant-abutment connection. These enhancements aim to achieve better biocompatibility, improved biomechanics, increased bone-implant contact surface area, and enhanced immunological response. This study aimed to investigate the influence of crown dimensions on stress distribution in the abutment screw during loading, utilizing finite element analysis (FEA). A comparative analysis of different dental implants from the same manufacturer was conducted to evaluate their biomechanical properties. The Von Mises analysis provided insights into the biomechanical behavior of these implants. The results indicate that an increase in both horizontal and vertical cantilever lengths can potentially elevate the risk of screw loosening and fatigue fracture. This can be attributed to the heightened stress values observed in the screw or other components, such as the abutment and fixture, respectively. These findings emphasize the importance of considering crown dimensions and their impact on stress distribution during implant design and placement to ensure optimal clinical outcomes and long-term stability.

Keywords
Implants, Implant Prosthesis, Molar Teeth, Finite Element Analysis (FEM), Von Mises analysis, Dental Stress Analysis

1. Introduction
A dental implant is widely accepted and effective for replacing lost teeth and restoring function, comfort, esthetics, speech, and tissue health. The selection of dental implants is primarily driven by the need to preserve the alveolar bone, which serves as the foundation for implant support. Intraosseous dental implants, as alloplastic materials, are surgically inserted into the residual alveolar ridge to function as prosthetic abutments [1].

However, it is essential to consider the mechanical aspects of dental implants to ensure their long-term success. Excessive stresses on the implant components can lead to complications and damage. The abutment, an integral part of the implant, is crucial in maintaining the prosthetic part or the superstructure. The superstructure, typically a metal framework, provides retention for removable prostheses or forms the framework of fixed partial dentures [2].

Among the components of dental implants, the abutment screw is of particular significance. It is the most accessible, reliable, and efficient means of fixing prosthetic components to the implant body. The abutment screw allows for easy retention on a small scale. However, high-stress levels can result in micro cracks in the surrounding bone, leading to bone resorption or mechanical failure of the implant or prosthetic components [3].

Unlike natural teeth, dental implants may not exhibit reversible signs or symptoms when experiencing complications such as bone resorption or restoration loosening. Abutment screw loosening serves as an indication of biomechanical stresses surpassing the assembly's tolerance threshold. Notably, implant crowns generally show minimal clinical signs other than fatigue and fracture. Consequently, dental clinicians face challenges in diagnosing and reducing stress levels applied to the supporting system [4].

Complications related to abutment and prosthetic screws, including loosening and fracture, can occur in implant prostheses. The prevalence of abutment screw fracture is relatively low compared to prosthetic screw fracture due to its larger diameter. However, abutment screw loosening is a common occurrence, affecting an average of 6% of implant prostheses. The risk of abutment screw loosening is directly correlated with the level of stress applied to the prosthesis. Cantilevers, which increase the load on the implant assembly, heighten the risk of screw loosening, particularly when the crown height attached to the abutment is increased [5].

Single-unit crowns demonstrate a higher rate of abutment screw loosening, which can lead to crestal bone loss. Cement-retained restorations require crown perforation to access the abutment screw in cases of loosening. Chronic screw loosening can be time-consuming and costly. Studies have shown that 6% to 20% of maxillary prostheses experience screw loosening within the first year of function. Factors such as occlusal imbalance, poor casting adaptation, and unequal forces can contribute to crown vibration during function, leading to screw loosening or fracture [6].

External forces exerted on the abutment screw significantly increase the risk of loosening. These forces, known as detaching or detorque forces, can cause screw loosening and are considered risk factors for implant fracture, crestal bone loss, and component fracture. When screws are correctly tightened and subjected to occlusal forces without any detaching force, they remain secure. However, if the external detach forces exceed the screw tightening forces, screw loosening occurs. Parafunctional habits, crown height, mastication dynamics, position in the dental arch, and opposing teeth are factors that increase stress on the implant and screw. Predictors of such conditions, including cantilevers, angulated forces, and poor occlusal schemes, should be considered [7].

The current study aims to evaluate the impact of crown dimensions on stress distribution in the abutment screw using three-dimensional finite element analysis (FEA). By analyzing the vertical and horizontal dimensions of the crown, the study seeks to provide insights into the biomechanical behavior of the abutment screw under load application. Understanding dental implants' mechanical and biomechanical aspects is essential for optimizing their design and ensuring their long-term stability and function. Finite element analysis is a valuable tool for assessing stresses and strains within dental implant components, offering useful insights into stress distribution and joint stability.

2. Materials and Methods

This study used a randomly selected computed tomography (CT) scan from Chamran Hospital in Shiraz. The scan involved patients with implants of different heights (11.5 mm, 10 mm, and 8.5 mm) and a diameter of 4 mm from the AnyRidge implant system by MegaGen in Daegu, Korea. The implants and straight abutments underwent 3D scanning using a 3D scanner equipped with an industrial camera and a light source. The resulting data, including information on the mandible, fixture, crown, and abutment geometry, were processed using software. Specific criteria were applied to design the teeth in 3D, including selecting cases of patients aged between 20 and 42 without systemic or bone diseases, non-smokers, and non-alcohol consumers. [8] Ultimately, a CT scan of a 22-year-old female was chosen for the study. Materialise Mimics Research version 21 software was used for the 3D design of the mandible, gingiva, and teeth. The files were imported, evaluated, and modified, and a mask for the bony segment was created based on specific Hounsfield unit values [9] (Figure 1).

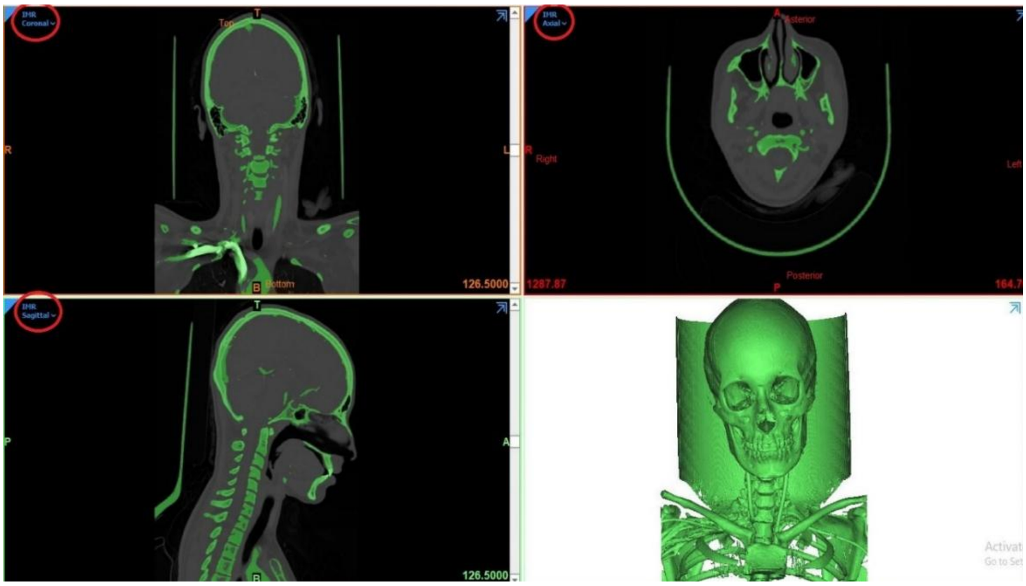


Figure 1. Axial, Coronal and Sagittal views and corrected directions

The Pre-design process focused on the maxilla and mandible, excluding surrounding areas and removing artefacts and noises. (Figure 2) Further refinement was done using 3-Matic Research version 13 software.

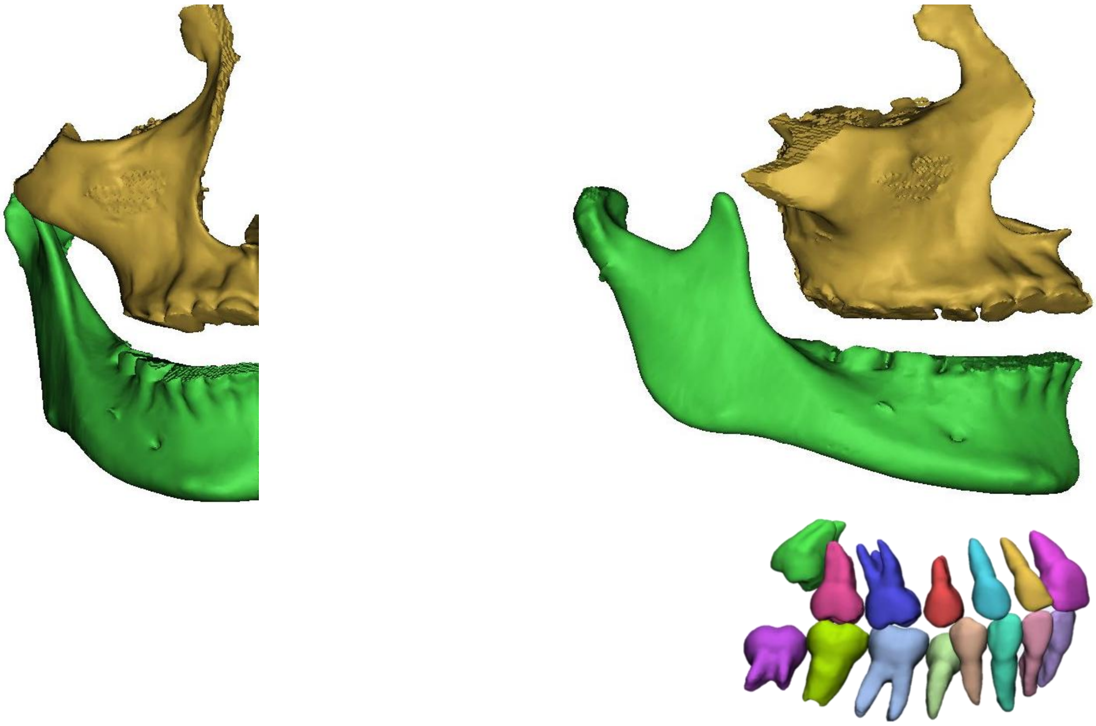


Figure 2. Pre-design of the maxilla mandible and teeth

The final models were obtained after aligning the teeth with the maxilla, mandible, and gingiva. (Figure 3)

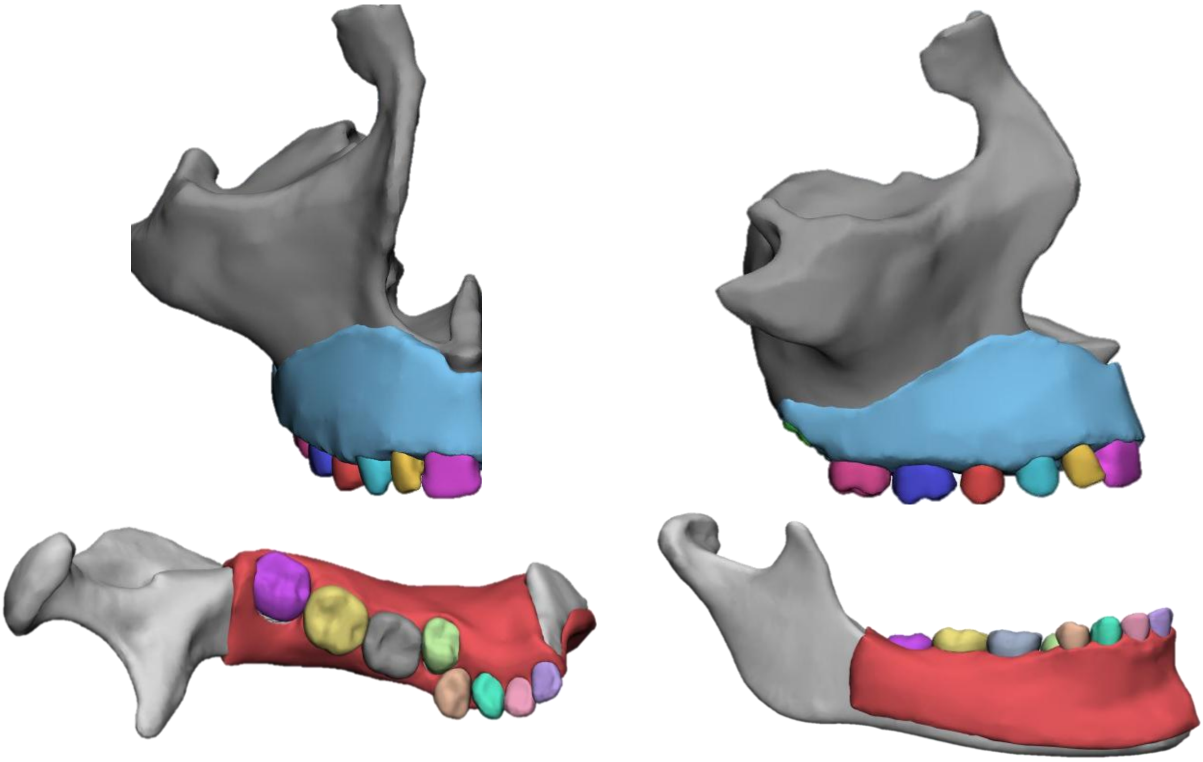


Figure 3. Design of the maxilla and mandible, teeth and gingiva

The designed bone had a D2 density and consisted of cortical and cancellous parts (Figure 4).

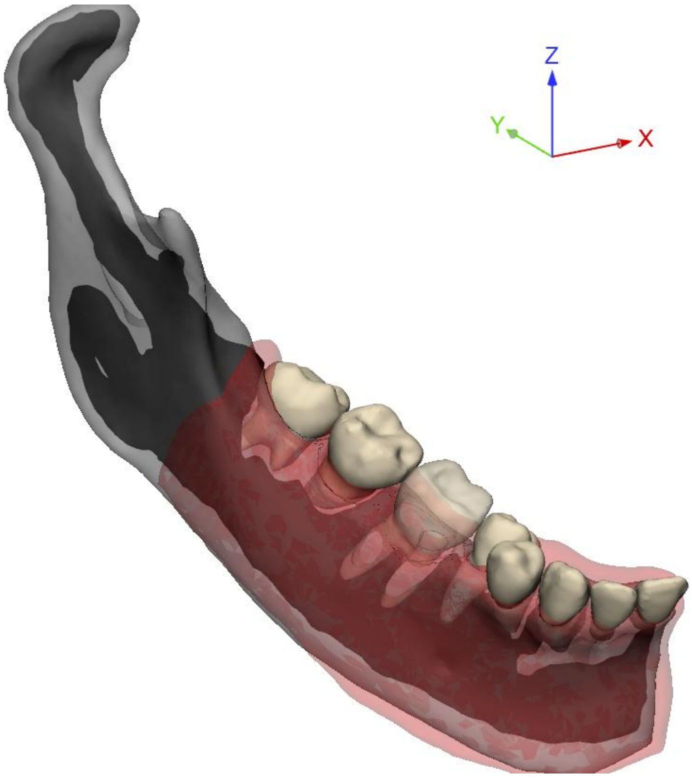


Figure 4. Design of the D2 density bone

The study aimed to assess the effect of loads on the implant-abutment interface and abutment screw loosening using finite element analysis (FEA). Megagen implants were used for modeling, as detailed geometric information about this implant system was unavailable. The implants, abutment screws, and abutments were scanned using a 3D scanner with an industrial camera and light source [10] (Figure 5).

Figure 5. Files extracted from the 3D scanner

Based on the obtained files and manufacturer information, specific implant models (ANYRIDGE XPEED FIXTURE 4.0 X 11.5MM) and post-EZ abutments were created using Solidworks software (Figures 6 and 7).

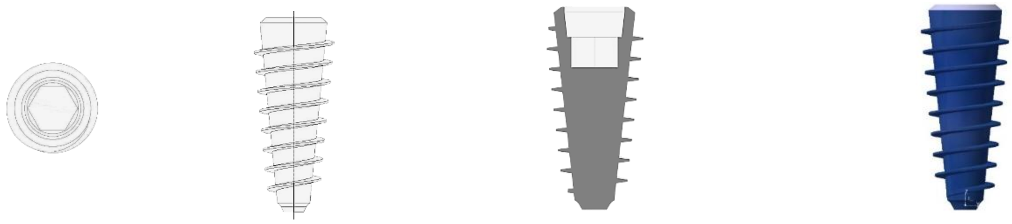
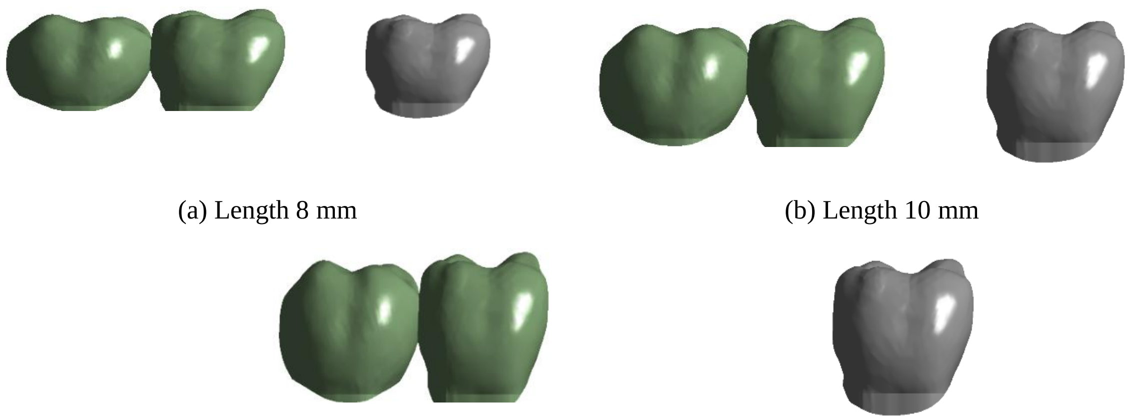


Figure 6. Simulated implant model

Figure 7. Simulated abutment

In the design, the implant-abutment contact area, where microorganisms can accumulate, was inwardly beveled by 0.5 mm. The implant was positioned at a 0.5-mm distance from the bone crest. A cement-retained prosthesis was designed, including a metal-ceramic coping with specific occlusal thickness and porcelain dimensions for teeth 6 and 7. The designs underwent refinement using different software programs (Figure 8).



(a) Length 8 mm

(b) Length 10 mm

(c) Length 12 mm

Figure 8. Simulation of the cement-retained prosthesis, including a metal-ceramic coping with specific occlusal thickness and porcelain dimensions for teeth 6 and 7

Considering the study variables, the crowns were designed with six horizontal and vertical cantilevers. The models were created using Mimics, 3-Matic, Solidworks, and ANSYS software. The models were then meshed using ANSYS R2 2020, with an average number of nodes and elements of 214354 and 223794, respectively. The mesh convergence study was conducted, and the results are presented in Figure 9(A). Changes were made in the meshing process to ensure accurate modeling with a stress change below 1% [11-13] (Figure 9, B, C).

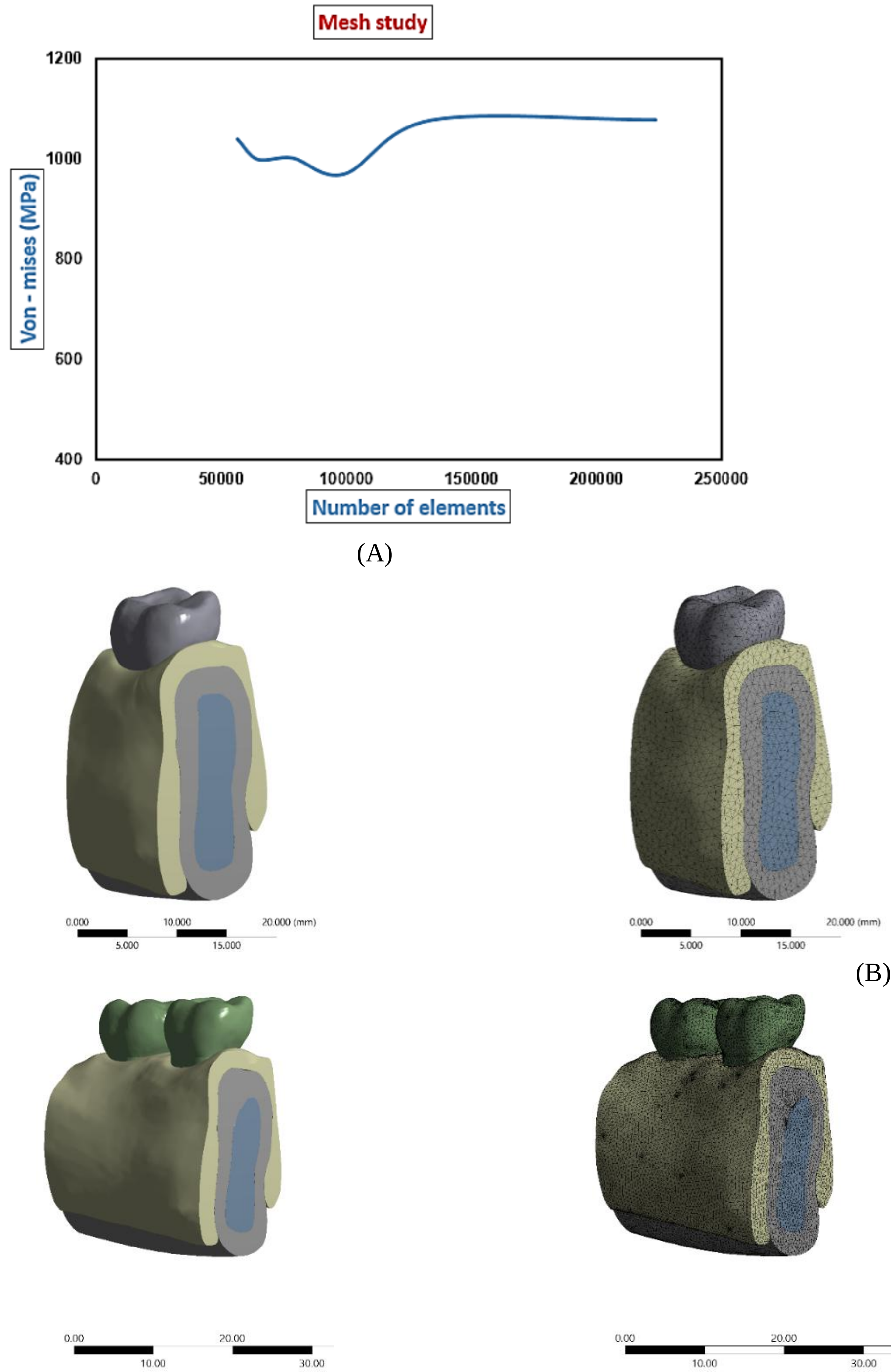


Figure 9. A) Mesh study, B) graphMeshed model for tooth 6, C) graphMeshed model for teeth 6 and 7

Two different cantilever lengths and three different heights of an implant with a 4-mm diameter and 11.5-mm height in a mandible with D2 bone density were subjected to static loading [14] (Table. 1).

Table 1. The force applied to the tooth

	Force along the Z axis (N)	Force along the Y-axis (N)
Implants and implant prosthesis	120	-20
Molars	150	-25
Preload	875	

Ultimately, six finite element models were analyzed. The apical part of the bone and the buccal and lingual surfaces were fixed before applying the loads (Figure 10).

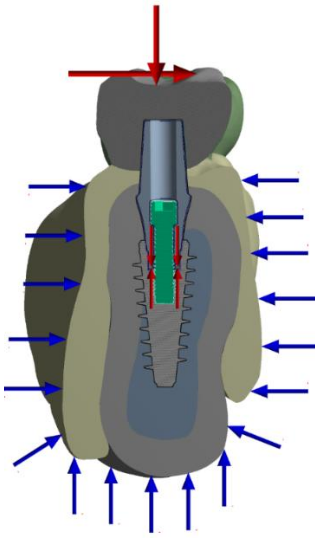


Figure 10. site of load application

Once the models were designed, the physical properties of the materials were added [15] (Table. 2).

Table 2. Modulus of elasticity and Poisson's ratio for the materials used for modeling

Material	Modulus of elasticity (MPa)	Poisson's ratio
Cortical bone	1.37×10^{10}	0.3
Cancellous bone	1.37×10^{10}	0.3
Mucosa	1×10^{7}	0.4
Acrylic resin	2.7×10^{9}	0.35
Titanium	1.17×10^{11}	0.33

The study evaluated the interaction effects of specific loads along different axes on the implant, implant crown, and cantilever at the sites of the first and second molars [16]. The preload applied was 850 N.

3. Results

The study aimed to evaluate the maximum von Mises stress values during the masticatory cycle by applying axial loads on prostheses and crowns with vertical and horizontal cantilevers. After the modeling process, the obtained data were carefully analyzed. The stress distribution was visualized using a color map, with warm colors indicating high-stress areas and cold colors representing low-stress regions. Stress values were reported in Pascals (Pa). The analysis of the images revealed that the stress experienced by all implant components remained below the yield strength of titanium (1020 MPa). This indicates no static degradation or plastic deformation of the implant system. Notably, the maximum stress distribution was observed at the fixture neck. The application of preload ensured an optimal mechanical connection between the implant and abutment through the abutment screw (Figure 11).

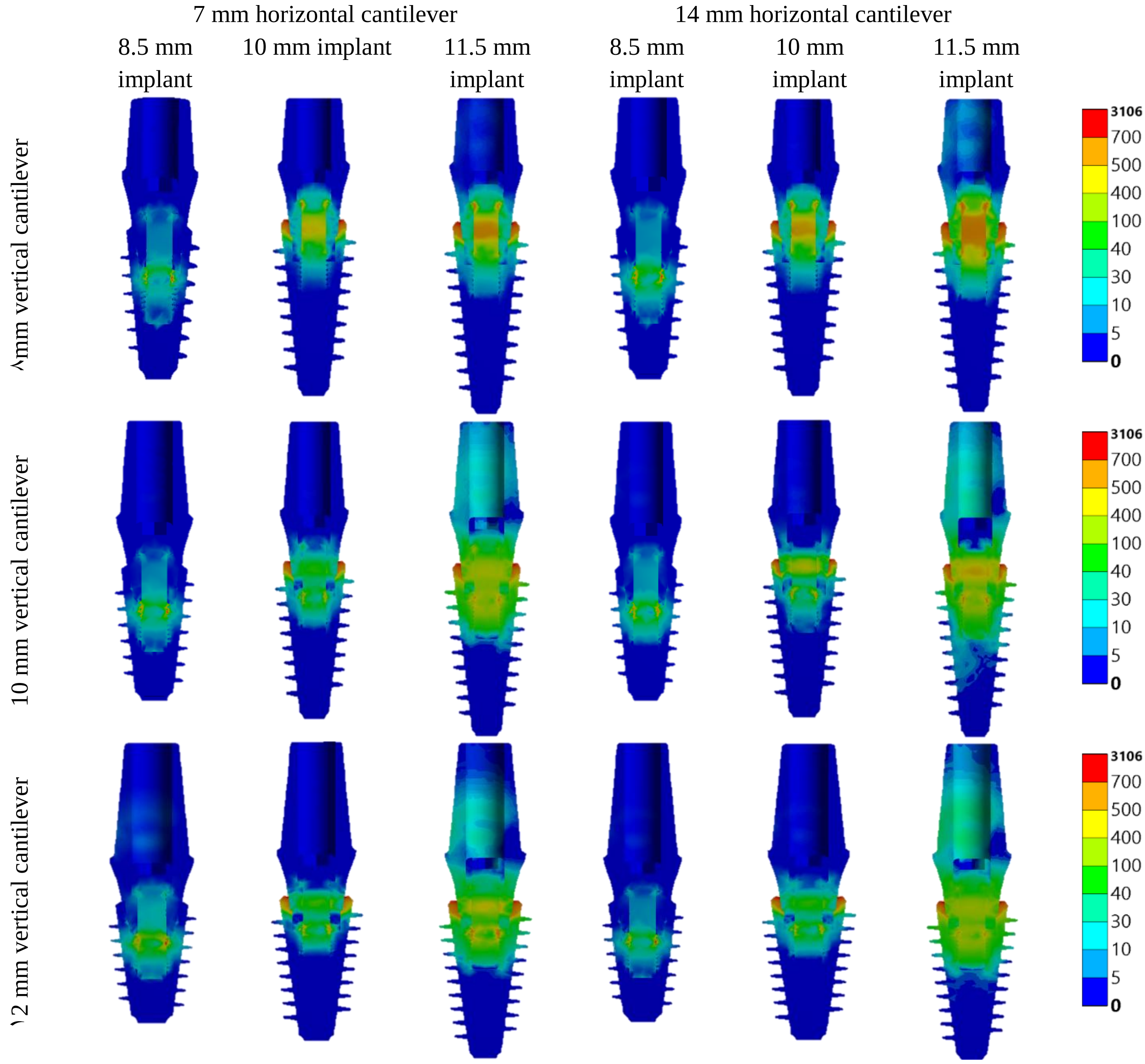


Figure 11. The pattern of von Mises stress distribution in the fixture implant system

Regardless of the crown geometry, the maximum stress in the abutment screws was consistently recorded in the screw body, the interface between the abutment screw body and screw access hole, and the abutment screw threads in the presence of each vertical cantilever (8, 10, 12 mm) and for implants with fixture heights of 8.5, 10, and 11.5 mm, an increase in horizontal cantilever length resulted in increased stress in the abutment screw. On the other hand, in implants with 7- and 14-mm horizontal cantilevers, the stress in the abutment screw decreased with an increase in vertical cantilever. Additionally, for implants with 7- and 14-mm horizontal cantilevers and 8-, 10-, and 12-mm vertical cantilevers, an increase in fixture height led to increased stress in the abutment screw. (Figures 12 and13)

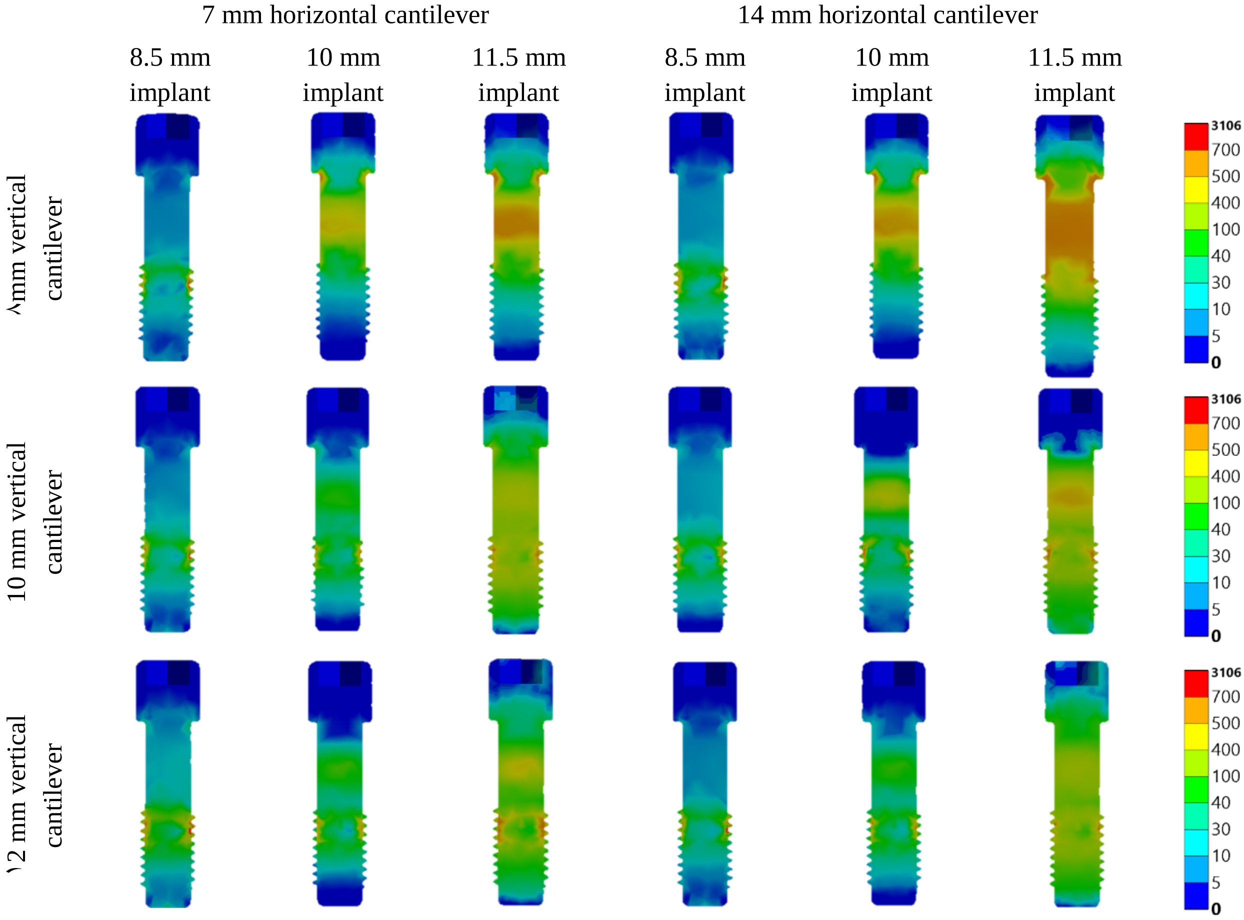


Figure 12. The pattern of von Mises stress distribution in the abutment screw

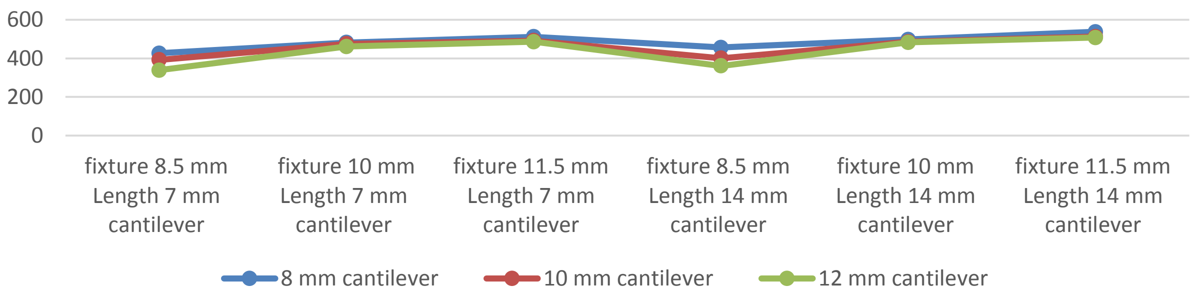


Figure 13. Graph of the comparison of von Mises stress distribution in the abutment screw

The fixture was crucial in transferring loads from the implant system to the bone. The fixture needed to be securely bonded to the bone to ensure effective load transfer. Higher stress distribution was observed at the fixture neck and the attachment points between the screw and fixture, resulting in a relatively round pattern of stress distribution in the cross-sectional view. Similar to the abutment screws, an increase in horizontal cantilever led to increased stress in the fixture for all three fixture heights (8.5, 10, and 11.5 mm) and vertical cantilevers (8, 10, and 12 mm). Likewise, an increase in vertical cantilever resulted in increased stress in the fixture for 7- and 14-mm horizontal cantilevers and an increase in fixture height led to increased stress in the fixture for implants with 7- and 14-mm horizontal cantilevers and 8-, 10-, and 12-mm vertical cantilevers (Figures 14 and15).

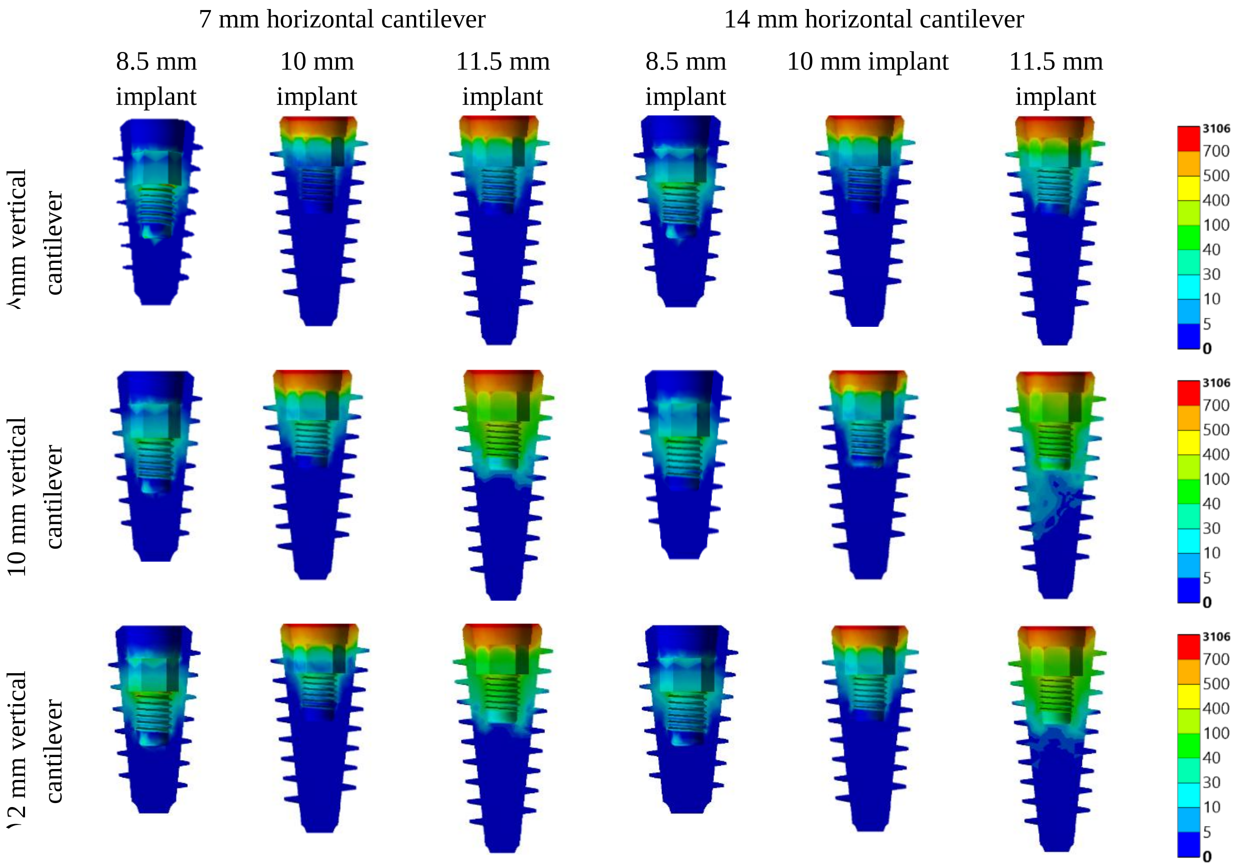


Figure 14. The pattern of von Mises stress distribution in the fixture

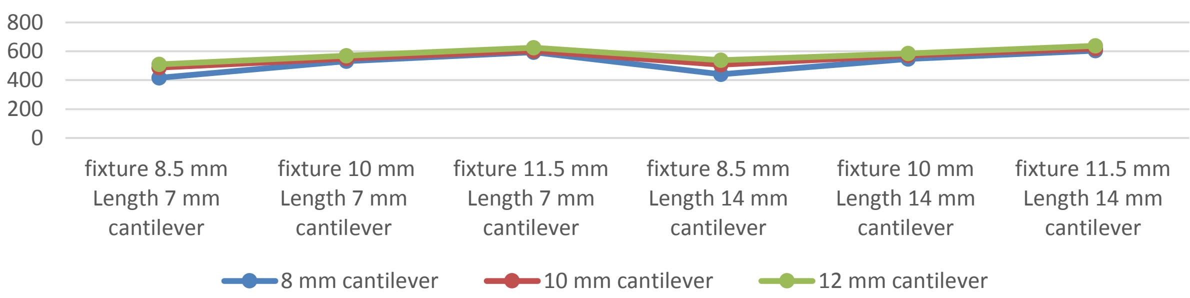


Figure 15. Graph of the comparison of von Mises stress distribution in the fixture

The abutment, which is responsible for withstanding variable masticatory forces, exhibited maximum stress at the abutment-fixture interface and the internal contact with the screw. Similar to the previous observations, an increase in horizontal cantilever led to increased stress in the abutment for all three fixture heights (8.5, 10, and 11.5 mm) and vertical cantilevers (8, 10, and 12 mm). Likewise, an increase in vertical cantilever resulted in increased stress in the abutment for 7- and 14-mm horizontal cantilevers and an increase in fixture height led to increased stress in the abutment for implants with 7- and 14-mm horizontal cantilevers and 8-, 10-, and 12-mm vertical cantilevers (Figures 16 and 17).

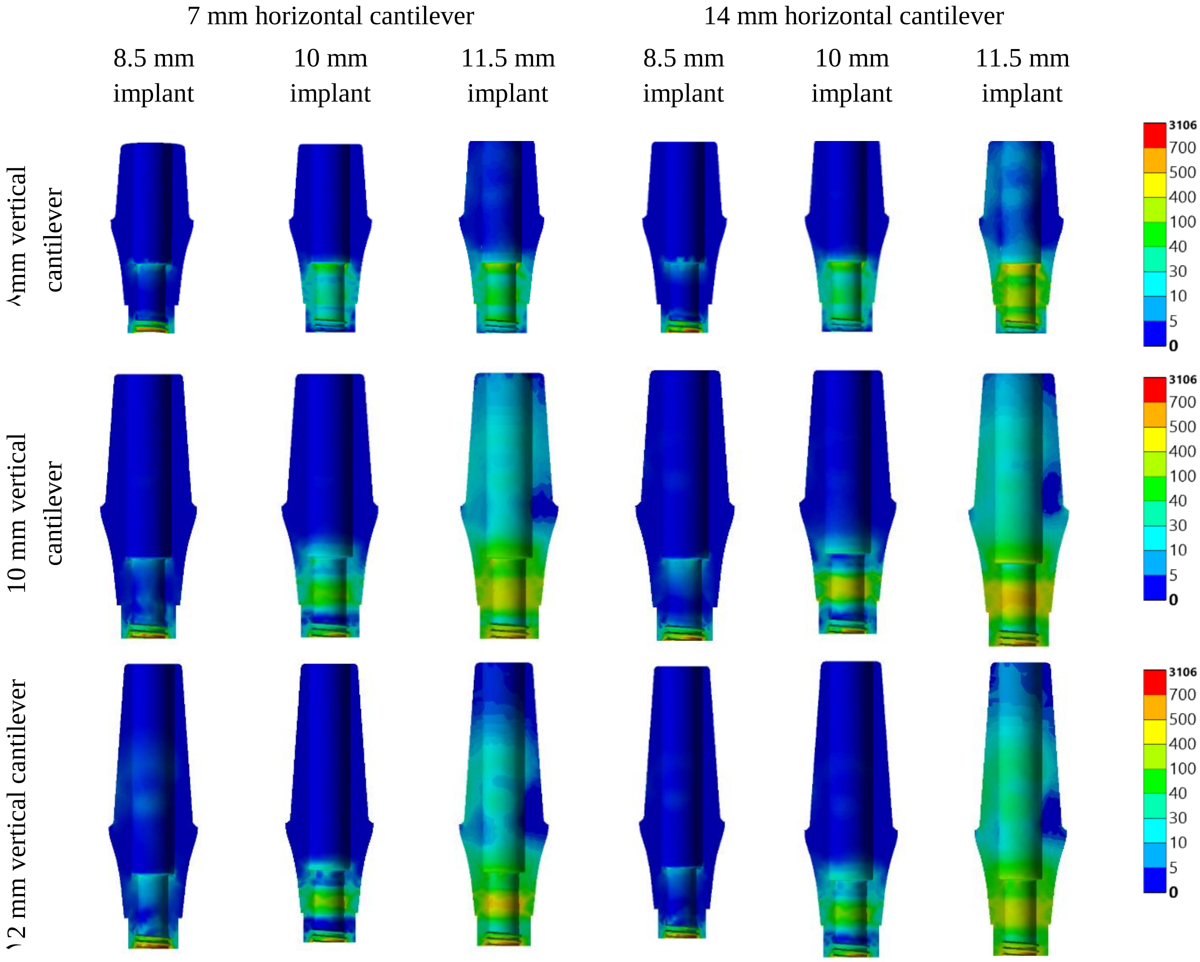


Figure 16. The pattern of von Mises stress distribution in the abutment

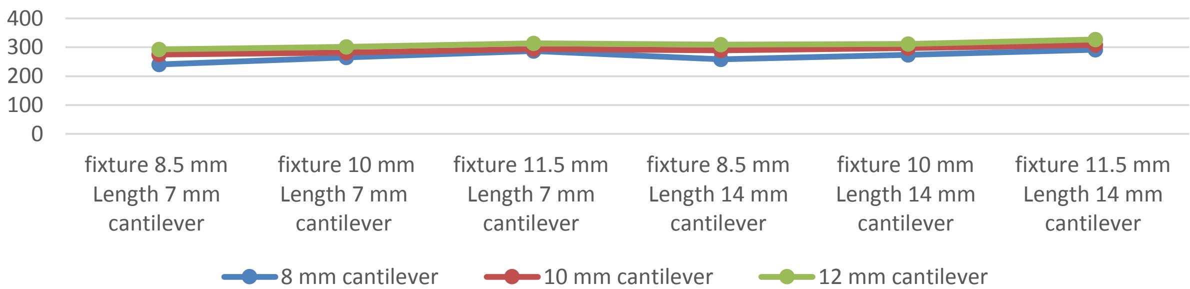


Figure 17. Graph of the comparison of von Mises stress distribution in the abutment

Table 3 provides a comprehensive comparison of stress distribution in different components of the implant assembly, highlighting the variations observed under various conditions (Table 3).

Table 3. Comparison of stress distribution (MPa) in different components of the implant assembly

14 mm horizontal cantilever			7mm horizontal cantilever				
11.5 mm implant	10 mm implant	8.5 mm implant	11.5 mm implant	10 mm implant	8.5 mm implant		
538	498	456	512	482	426	abutment screw	10 mm vertical cantilever
291	274	258	287	265	241	abutment	
604	547	441	594	531	417	fixture	
513	486	401	492	475	392	abutment screw	10 mm vertical cantilever
308	297	289	294	281	275	abutment	
618	572	506	603	548	486	fixture	
508	483	361	486	461	338	abutment screw	12 mm vertical cantilever
327	311	309	314	301	293	abutment	
637	584	538	624	569	510	fixture	

4. Discussion

Bone loss following dental implant treatment can present significant challenges. Insufficient available bone may limit the placement of implants with optimal height and diameter. Additionally, systemic diseases, old age, and financial constraints can make complex surgical procedures like bone grafting impractical. In such cases, dental implants with vertical and horizontal cantilevers may be utilized to avoid needing advanced removable prosthetic or surgical treatments. However, it is essential to note that increased vertical and horizontal cantilevers can exert more significant stress on the fixture, potentially leading to complications such as crestal bone loss and prosthetic issues like screw fracture and loosening. These complications can be problematic for both patients and dental clinicians, underscoring the need to identify contributing factors and develop strategies for prevention. The success of implant treatment depends not only on Osseo integration but also on biomechanical factors. The geometry of the implant assembly and the magnitude of vertical and horizontal cantilevers can influence the stress applied to the implant and the supporting bone through the lever mechanism. Larger crown dimensions combined with shorter implants may result in higher stress levels. The crown/implant height ratio is more excellent with shorter implants, and it is important to anticipate potential biomechanical complications in these cases. However, previous studies have shown that short implants can be clinically successful regardless of their crown/implant height ratio. [17-20] The material of the abutment screw can also impact preload. Gold alloy screws have higher tensile and yield strength values compared to conventional titanium screws, enabling the achievement of higher preload with gold alloy screws. For consistency, the present study employed the same type of screw in all three implant heights, despite variations in vertical and horizontal cantilever lengths. [21] Finite Element Analysis (FEA) is a cost-effective method for assessing stress distribution in implant system components. It provides a simple approach to evaluating complex biomechanical systems. However, to ensure reliable modeling, several parameters must be considered, including the precise mechanical properties of the implant system, implant system geometry, abutment screw preloading, and reverse engineering.

4.1 Effects of horizontal cantilever

The results revealed that increasing the horizontal cantilever resulted in increased stress in the abutment screw (specifically in the screw body, interface between the screw body and screw hole, and abutment screw threads), abutment (at the abutment-fixture contact and internal contact with the screw), and fixture (around the fixture neck and at the contact point between the screw and fixture).

4.2 Effects of the vertical cantilever

The findings demonstrated that increasing the vertical cantilever led to decreased stress in the abutment screw (in the screw body and contact with the abutment) for 7- and 14-mm horizontal cantilevers while increasing stress in the abutment (at the abutment-fixture contact and internal contact with the screw) and fixture (around the fixture neck and at the contact point between the screw and fixture) for all three fixture heights (8.5, 10, and 11.5 mm).

4.3 Effects of fixture height

Applying loads to implants with 7- and 14-mm horizontal cantilevers and 8-, 10-, and 12-mm vertical cantilevers resulted in increased stress in the screw (in the screw body and contact area between the screw body and screw hole, as well as the screw threads), the abutment (at the abutment-fixture contact and internal contact with the screw), and fixture (around the fixture neck and at the contact point between the screw and fixture) with an increase in fixture height.

Overall, it can be concluded that the minimum screw stress was observed in implants with 8.5 mm fixture height, 7 mm horizontal cantilever, and 12 mm vertical cantilever. Conversely, the maximum screw stress was noted in implants with 11.5 mm fixture height, 14 mm horizontal cantilever, and 8 mm vertical cantilever. It is worth noting that a study by Oyar et al. [22] suggested that the length of the horizontal cantilever and posterior implant inclination influenced the load distribution pattern. They found that increasing the horizontal cantilever decreased stress in posterior implants with a distal inclination, which differs from the present findings regarding the effect of horizontal cantilever on stress distribution in implant components. These variations in results may be attributed to differences in methodologies, study designs, implant brands with varying shapes and angles, load application angles, and the influence of other parameters on stress generation.

5. Conclusion

The present study revealed that the fixture neck experienced the highest stress levels within the implant system. Additionally, the screw body, contact area between the screw body and screw hole, and screw threads exhibited maximum stress in the screw component. The variation in vertical and horizontal cantilevers resulted in fatigue of the abutment screw, leading to the screw loosening and affecting the contact between the screw and the abutment. Under the application of loads, stress accumulates at the fixture-abutment, fixture-screw, and abutment-screw interfaces. It is important to note that the abutment screw in single crowns demonstrated higher stress tolerance compared to crowns involving two teeth.

6. References

[1] Arunyanak, S. P., Harris, B. T., Grant, G. T., Morton, D. and Lin, W. S. 2016. Digital approach to planning computer-guided surgery and immediate provisionalization in a partially edentulous patient. Journal of prosthetic dentistry. 116(1):8-14. doi: 10.1016/j.prosdent.2015.11.023.

[2] Dubova, L. V., Shlyk, A. D., Romankova, N. V., Sokolova, M. S. and Maximov, G. V. 2022. Analysis of the condition of the fixing screw for different types of implant and abutment connections. Actual problems in dentistry. 17(4):13-17. doi: 10.18481/2077-7566-21-17-4-13-17.

[3] MousaviJazi, M., Naderan, A., Ebrahimpoor, M. and Sadeghipoor, M. 2013. Association between psychological stress and stimulation of inflammatory responses in periodontal disease. Journal of dentistry. 10(1):103-111.

[4] Patil, P. G. 2011. A Technique for Repairing a Loosening Abutment Screw for a Cement-Retained Implant Prosthesis. Journal of Prosthodontics. 20(8): 652-655. doi: 10.1111/j.1532-849X.2011.00748.

[5] Goodacre, B. J., Goodacre, S. E. and Goodacre, C. J. 2018. Prosthetic complications with implant prostheses. European Journal of Oral Implantology. 11(1): 27-36.

[6] Siamos, G., Winkler, S. and Boberick, K. G. 2002. The relationship between implant preload and screw loosening on implant-supported prostheses. Journal of Oral Implantology. 28(2): 67-73. doi: 10.1563/1548-1336(2002)028<0067:TRBIPA>2.3.CO;2.

[7] Volkmann, H., Rauch, A., Koenig, A. and Schierz, O. 2022. Pull-off Force of Four Different Implant Cements Between Zirconia Crowns and Titanium Implant Abutments in Two Different Abutment Heights. International Journal of Periodontics & Restorative Dentistry, 42(3):67-74. doi: 10.11607/prd.4926.

[8] Calado, G. 2020. Development of methodologies for raman spectral analysis of human saliva for detection of oral cancer. doi: 10.21427/6hty-gf29.

[9] Tahavvor, A. R., Zarrin Chang, P., Abadi Iranagh, S. and Heidari, S. 2017. Numerical simulation of realistic human lumbar spine model under compressive force, axial rotation and lateral bending loads. Modares Mechanical Engineering. 16(11):54-60. dor: 20.1001.1.10275940.1395.16.11.27.0.

[10] Tahavvor, A. R., Zarrinchang, P. and Heidari, S. 2015. Numerical simulation of turbulent airflow in a human upper respiratory system. Modares Mechanical Engineering. 14(15):267-272.

[11] Rastitalab, A., Khajepour, S., Dehghani, J., Afsari, A. and Heidari, S. 2021. Evaluating the Stability of the Fractured Bone Implanted with Titanium Elastic Nails in C and S Configurations. Journal of Hunan University Natural Sciences. 48(9):349-358.

[12] Rastitalab, A., Khajehpour, S., Afsari, A., Heidari, S. and Dehghani, J. 2021. Mechanical stability of RCSed and ECAPed intramedullary 316L stainless steel nails in the treatment of diaphyseal bone fractures. Journal of Modern Processes in Manufacturing and Production. 10(4):37-49. dor: 20.1001.1.27170314.2021.10.4.3.8.

[13] Rastitalab, A., Khajehpour, S., Afsari, A., Heidari, S. and Dehghani, J. 2021. An investigation of using RCS-processed intramedullary stainless steel 316L nail in the treatment of diaphyseal bone fractures. Journal of Modern Processes in Manufacturing and Production. 10(3):59-75. dor: 20.1001.1.27170314.2021.10.3.8.1.

[14] Sudharshan, R. 2012. Comparative evaluation of the microgap of premachined Titanium and Zirconia abutments at the implant-abutment interface before and after cyclic loading: An in Vitro study. Doctoral dissertation. Ragas Dental College & Hospital, Chennai.

[15] Assunção, W. G., Barão, V. A. R., Gomes, É. A., Delben, J. A. and Ribeiro, R. F. 2012. FEA in Dentistry: A Useful Tool to Investigate the Biomechanical Behavior of Implant Supported Prosthesis. Finite Element Analysis: From Biomedical Applications to Industrial Developments. 3:57-80. doi:10.5772/38686.

[16] Pourabbas, B., Emad, J., Dehghani, J., Heidari, S. and Vosoughi, A. R. 2023. Mechanical evaluation of the effect of the rod to rod distance on the stiffness of uniplanar external fixator frames. Musculoskeletal surgery.107(4):1-7. doi:10.1007/s12306-023-00782-1.

[17] Fabris, D., Moura, J. P., Fredel, M. C., Souza, J. C., Silva, F. S. and Henriques, B. 2022. Biomechanical analyses of one-piece dental implants composed of titanium, zirconia, PEEK, CFR-PEEK, or GFR-PEEK: Stresses, strains, and bone remodeling prediction by the finite element method. Journal of Biomedical Materials Research Part B: Applied Biomaterials. 110(1):79-88. doi:10.1002/jbm.b.34890.

[18] Alemayehu, D. B. and Jeng, Y. R. 2021. Three-dimensional finite element investigation into effects of implant thread design and loading rate on stress distribution in dental implants and anisotropic bone. Materials. 14(22):6974-6998. doi: 10.3390/ma14226974.

[19] Kligman, S., Ren, Z., Chung, C. H., Perillo, M. A., Chang, Y. C., Koo, H., Zheng, Z. and Li, C. 2021. The impact of dental implant surface modifications on osseointegration and biofilm formation. Journal of clinical medicine. 10(8):1641. doi:10.3390/jcm10081641.

[20] Tolstunov, L., Hamrick, J. F. E., Broumand, V., Shilo, D. and Rachmiel, A. 2019. Bone augmentation techniques for horizontal and vertical alveolar ridge deficiency in oral implantology. Oral and Maxillofacial Surgery Clinics. 31(2):163-191. doi:10.1016/j.coms.2019.01.005.

[21] Vinhas, A. S., Aroso, C., Salazar, F., López-Jarana, P., Ríos-Santos, J. V. and Herrero- Climent, M. 2020. Review of the mechanical behavior of different implant–abutment connections. International Journal of Environmental Research and Public Health. 17(22):8685-8705. doi: 10.3390/ijerph17228685.

[22] Oyar, P., Durkan, R. and Deste, G. 2021. The effect of the design of a mandibular implant-supported zirconia prosthesis on stress distribution. The Journal of prosthetic dentistry. 125(3): 502.e1-502.e11. doi: 10.1016/j.prosdent.2020.05.027.