JPTM
http://journal.unnes.ac.id

EFFECT OF IMPLANT DIAMETER ON BIOMECHANICAL RESPONSE IN ANTERIOR MAXILLA IMPLANT-SUPPORTED BRIDGE: A 3D FEA

Beni Aji Subekti[1], Kriswanto[1], Febri Budi Darsono [1], Rahmat Doni Widodo [1], J. Jamari [2], Athanasius Priharyoto Bayuseno[2]

[1] Mechanical Engineering, Faculty of Engineering, Semarang State University, Semarang, Central Java, Indonesia.
[2] Mechanical Engineering, Faculty of Engineering, Diponegoro University, Semarang, Central Java, Indonesia.
Email: beniajis009@students.unnes.ac.id

Abstract

This study evaluates the effect of varying implant diameters on stress, strain, and factor of safety (FOS) distribution in an anterior implant-supported bridge using finite element analysis (FEA). Unlike most previous studies, this research focuses on a complete implant-supported bridge and examines the abutment and screw components separately. A three-dimensional model of the anterior maxilla was reconstructed from CBCT data of a healthy adult male using Inventor and ANSYS software. Three implant diameters, 3.75 millimeters, 4.3 millimeters, and 5.0 millimeters, were simulated under a static occlusal load of 204.1 newtons applied at a 45-degree angle. The 5.0 millimeter implant reduced maximum stress by 23.7 percent on the abutment and 33.9% on the screw compared to 3.75 millimeters, and yielded the highest FOS value of 8.00. Strain values also decreased and remained below the elastic limit, indicating no plastic deformation. Larger diameters improved biomechanical stability and load distribution. Future studies should include dynamic loading and patient-specific modeling.

Key words: Finite element analysis, Fixture diameter, Anterior maxilla, Abutment, Screw

INTRODUCTION

Maxillary dental rehabilitation represents a significant clinical challenge in modern dental practice. The anterior teeth, particularly the central and lateral incisors, not only play a crucial role in basic functions such as cutting food and assisting with phonetic articulation, but also significantly influence facial aesthetics and patient self-confidence (Budală et al., 2023; Fekonja, 2022). The presence of anterior teeth significantly influences facial proportions and social perception of appearance, making them a crucial aspect of restorative dental treatment, focusing on a multidisciplinary approach (Alshamri et al., 2023). Therefore, any rehabilitation effort in this area requires high precision, from a biomechanical perspective and an aesthetic and psychosocial perspective. Various clinical factors can cause the problem of anterior tooth loss, the most common being untreated chronic periodontal disease and direct trauma from accidents or sports activities (Biagi et al., 2024; Y. Wang et al., 2025). This condition has a significant impact, especially on individuals of productive age. Based on recent epidemiological data, approximately 46.7% of individuals aged 18–35 years experience the loss of one or more teeth in the maxillary anterior region (Karthiga Devi G et al., 2022). The growing need for evidence-based clinical procedures highlights the importance of anterior tooth replacement, designed for functionality, aesthetic appeal, and long-term stability. Removable dentures and implant-supported bridges are the most common tooth replacement methods. Although removable dentures are still used in selected cases, implant-supported bridges have emerged as a superior option due to their improved mechanical reliability, aesthetics relative to natural teeth, and cost-effectiveness over time (Bandiaky et al., 2022). These fixed prostheses also offer the advantages of minimally invasive procedures and better preservation of peri-implant soft tissue, ultimately resulting in greater patient comfort and satisfaction (Losenická et al., 2021). Findings from Takahashi et al. (2022) further support this preference, demonstrating a success rate of 99% at five years and 98.1% at ten years for implant-supported bridges, confirming their high long-term effectiveness in anterior tooth restoration.

An implant-supported bridge consists of several key elements that work together to restore tooth function, such as a fixture anchored in the bone, an abutment that serves as an intermediate connector, a screw that provides mechanical retention, and a crown that completes the prosthetic structure. These components function synergistically to transfer occlusal loads evenly. The fixture's diameter is crucial in establishing primary stability among these components. Research by Huang et al. (2023) showed that the implant diameter expands the contact area with the bone, which increases initial stability and reduces stress concentrations in the surrounding bone tissue. However, failures during the installation process can still occur. In some cases, the most common failure in dental implant installation is loosening of screws already installed on the implant. This is supported by research by Lee et al. (2020) who observed that 78.3% of screw loosening cases occurred in the anterior maxilla, primarily due to off-axis loads transmitted through the palatal aspect of the anterior teeth. A study by Szajek and Wierszycki (2023) emphasized that the geometric configuration of the fixture, especially its diameter, significantly influences the risk of loosening. Thus, selecting the correct fixture diameter is crucial for achieving biomechanical stability in anterior implant applications.

In addition to structural design, the implant material's physical and mechanical properties substantially influence the prosthesis's long-term outcome. The material must withstand repeated masticatory forces, resist fatigue degradation, and maintain compatibility with the surrounding biological structures. Titanium and its alloy Ti-6Al-4V are the most widely adopted materials due to their exceptional strength, wear resistance, and elastic modulus similar to cortical bone (Nicholson, 2020). Research by Nicholson (2020) revealed that Ti-6Al-4V has a wear resistance exceeding 89% and maintains a clinical survival rate of 97–99% over ten years, reinforcing its reputation for durability and functional life. These advantages are significant in high-stress areas such as the anterior maxilla, where mechanical demands are significant.

Finite Element Analysis (FEA) has become a widely used and reliable technique for evaluating the biomechanical behavior of dental implants (Falcinelli et al., 2023). FEA facilitates the simulation of stress distribution, deformation patterns, and overall structural performance under various loading conditions. In clinical and research settings, FEA is widely used to identify failures, optimize prosthetic designs, and assess implant response under bone density variations or angular loading (Byun et al., 2022). One of the fundamental parameters in FEA is the Factor of Safety (FOS), which compares the allowable stress of a material with the actual stress experienced under functional loads (Budynas & Nisbett, 2015). By incorporating FOS into simulation models, clinicians and researchers can estimate the probability of mechanical failure, thereby ensuring implant-supported restorations' structural safety and long-term reliability. A study by Kriswanto et al. (2025) showed that using FOS, von Mises stress, and maximum displacement can evaluate material combinations and implant component designs. Furthermore, a systematic review by Sayed et al. (2025) covering studies from 2012 to 2022 showed that increasing implant diameter consistently decreased von Mises stress and peri-implant bone strain. González-Mederos et al. (2025) study corroborated these findings by concluding that larger implant diameters improved load distribution and reduced the risk of stress concentrations. Although most studies have not explicitly addressed anterior implants or implant-supported bridges, the biomechanical principles related to implant diameter remain relevant for application to various locations within the oral cavity.

Although Finite Element Analysis (FEA) has been widely used in biomechanical studies of dental implants, most existing studies still focus on the analysis of single implant systems and variations in fixture diameters in general, without in-depth exploration of complex configurations such as implant-supported bridges in the anterior maxillary region. Anterior implant cases have different biomechanical characteristics, as the anterior bone generally has a lower density and is subjected to occlusal forces that vary in direction and intensity. Moreover, important components such as abutments and abutment screws are often critical points in stress and strain distribution, but have not been analyzed separately and systematically in the context of anterior bridge systems. Therefore, this study aims to fill this gap by conducting FEA-based numerical simulations to evaluate the effect of variations in implant fixture diameters of 3.75 mm, 4.3 mm, and 5 mm on the overall mechanical response of the system. The simulations were conducted considering loading conditions that represent actual clinical situations, including the direction of occlusal forces and tooth-to-tooth contacts. The parameters analyzed included von Mises stress, strain, and the Factor of Safety (FOS) of the abutment components and abutment screws, which are considered important in predicting the system's resistance to mechanical failure. With this approach, the study is expected to significantly contribute to understanding the role of implant diameter on the biomechanical performance of anterior bridge systems, while offering a scientific basis for safer and more effective implant planning in the anterior maxillary region.

METHODS

The bone modeling in this study was based on three-dimensional radiographic imaging data obtained via cone beam computed tomography (CBCT) of a healthy 21-year-old adult male subject, as shown in Figure 1a. CBCT provides highly accurate and detailed anatomical visualization, particularly of the anterior maxillary region, allowing for precise extraction of bone morphometric parameters. Critical information such as cortical thickness, length, and width of the anterior bone was obtained using RainBow Viewer software, which supports high-accuracy digital measurements. The measurements showed that the anterior maxillary length was 23 mm, with an average cortical bone thickness of 1.5 mm. These morphometric parameters were then used as a reference in the bone modeling process using CAD Inventor software, which was designed to resemble the original anatomical shape and dimensions of the CBCT reconstruction (Figure 1b). Based on the bone quality classification according to CARL E. MISCH (2008), the resulting bone model was categorized as type D3 because it was located in the anterior maxillary region, which generally has moderate to low bone density.

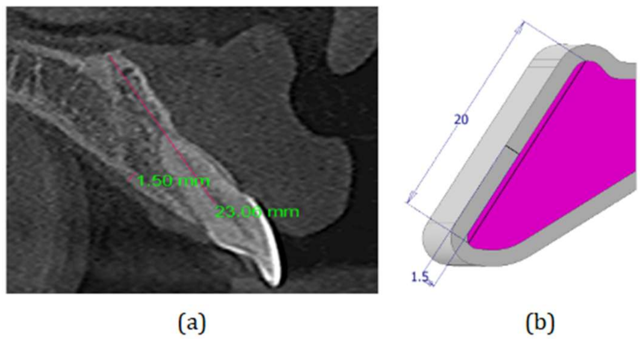

Figure 1. (a) CBCT anterior maxilla (b) 3D Inventor tulang anterior maxilla

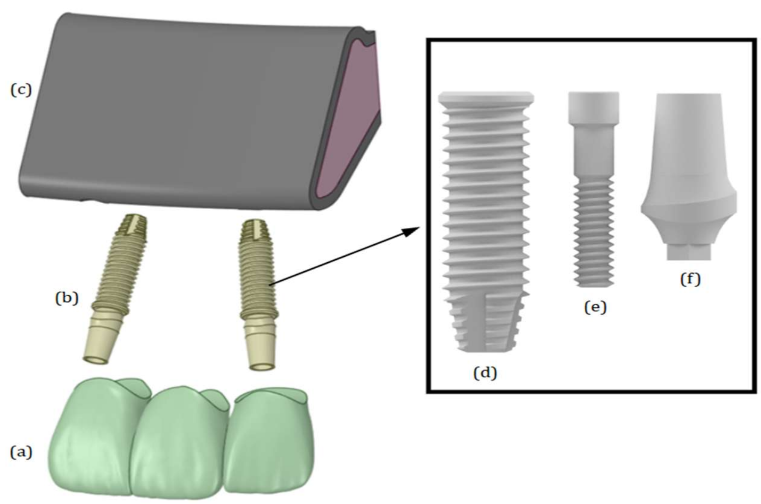

Figure 2. Components of the Dental Implant FE Model: (a) Crown bridge; (b) Full Implant; (c) Bone; (d) Fixture; (e) Screw; (d) Abutment

The structure of the implant bridge system used consists of four main components, namely fixtures, abutments, screws, and crowns, as shown in Figure 2. The fixture design refers to Nobel Biocare's commercial cone joint system, with a V-shaped thread profile commonly used in clinical practice due to its ability to produce optimal primary stability. The abutments used are esthetic types from the same system and are specifically designed to meet the needs of anterior restorations that demand high aesthetics. The screw components are also derived from the Nobel Biocare system to maintain compatibility and joint strength. The diameter fixture variations used in biomechanical simulations consist of 3.75 mm, 4.3 mm, and 5 mm. At the same time, other dimensions are kept constant, including a fixture length of 13 mm, a thread pitch of 0.6 mm, and a thread depth of 0.3 mm. The abutment size is adjusted based on the official catalog, with a 3.75 mm fixture attached to a smaller abutment. The crown model was derived from a modified OJRICK Surface design (OJRICK, 2025). The design was then converted to a solid state using Geomagic Design X software for FEA simulation purposes.

In the simulation process, all interfaces between components were modeled as bonded contacts to simplify mechanical interactions, although this approach has limitations in representing possible micro-movements between surfaces. The initial torque (preload) on the screw was not included in the modeling because the simulation was performed under the assumption of full osseointegration, following the approach used by González-Mederos et al. (2025). In addition, the numerical modeling structure in this study refers to the method developed by Kriswanto et al. (2025), which has been numerically validated through comparison with the Talmazov model. Despite implant location and clinical configuration differences, the applied FEA approach remains relevant as a methodological reference in biomechanical analysis. The finite element method (FEM) allows modeling of complex structures by discretizing the domain into small, interconnected elements. It can efficiently estimate the distribution of stresses, strains, and displacements under various loading conditions (Daryl L. Logan, 2017). With its ability to represent the structural response to loads in detail, FEM has become an important tool in modern engineering simulations prior to experimental testing. Mathematically, this analysis is based on the law of force balance formulated as a system of matrix equations, as shown in Equation (1).

$$\{F\} = [K]\{d\} \qquad (1)$$

In equation (1), {F} is the nodal force vector, [K] is the global stiffness matrix, and {d} is the nodal displacement vector. This equation is formulated based on the principle of force equilibrium in a linear elastic system.

In the numerical analysis stage using FEA, this study identified von Mises stress as the main parameter calculated based on the stress components in the XYZ direction, as shown in equation (2).

Equation (2) is a three-dimensional von Mises stress formulation calculated based on the normal stress components (σ_x, σ_Y, σz) and shear stress components (τ_{xY}, $\tau_Y z$, τz_x). This von Mises stress (σ') represents the equivalent stress used to evaluate the yield potential of a material based on the energy distortion criterion (Budynas & Nisbett, 2015).

$$\sigma' = \frac{1}{\sqrt{2}}\left[\left(\sigma_x - \sigma_y\right)^2 + \left(\sigma_y - \sigma_z\right)^2 + (\sigma_z - \sigma_x)^2 + 6\left(\tau_{xy}^2 + \tau_{yz}^2 + \tau_{zx}^2\right)\right]^{1/2} \qquad (2)$$

This study also considers the von Mises equivalent strain as one of the analysis parameters, with calculations referring to the formulation applied by (ANSYS, 2024) equation (3)

The equivalent von Mises strain ($\varepsilon_e q$) is calculated from the combination of the normal strain components (ε_x, ε_Y, εz) and shear strain components (γ_{xY}, $\gamma_Y z$, γz_x). This value is used to evaluate the level of multiaxial strain in the material and determine whether plastic strain exceeding the elastic limit has occurred.

$$\varepsilon_{eq} = \frac{1}{\sqrt{2(1+\nu')}}\left[\left(\varepsilon_x - \varepsilon_y\right)^2 + \left(\varepsilon_y - \varepsilon_z\right)^2 + (\varepsilon_z - \varepsilon_x)^2 + \frac{3}{2}\left(\gamma_{xy}^2 + \gamma_{yz}^2 + \gamma_{xz}^2\right)\right]^{1/2} \qquad (3)$$

In addition to calculating von Mises stress and strain, this study also evaluated the factor of safety (FOS) value to assess the structure's safety limits against mechanical failure. The FOS calculation refers to equation (4) in (Budynas & Nisbett, 2015).

Equation (4) shows that the design safety factor (n_d) is the ratio between the material strength (S) to the actual working stress, both normal stress (σ) and shear stress (τ), to assess whether the structure is in a safe condition against failure.

$$n_d = \frac{\text{loss-of-function strength}}{\text{allowable stress}} = \frac{S}{\sigma\ \ (\text{or } \tau)} \qquad (4)$$

In this study, modeling was performed using ANSYS Workbench 2023 R1 software, with the initial geometry designed in Autodesk Inventor and then exported for numerical simulation purposes. The element used was a 10-node tetrahedral type, which is considered suitable for complex three-dimensional stress analysis. The element size was set at 0,67 mm to balance the accuracy of results and computational time efficiency. The results were validated through a convergence test by comparing the differences in von Mises stress values between models, and a maximum tolerance was set at 5%, as shown in Table 1. Mesh quality was also analyzed based on the skewness value, with an average result of 0.510 ± 0.001, which is still classified as sufficient. The final modeling results produced a total number of nodes of 241,263, 249,745, and 291,495 for equipment diameters of 3.75 mm, 4.3 mm, and 5.0 mm, respectively, with a total number of elements of 141830, 147319, and 171755. One of the mesh results is illustrated in Figure 3 as follows.

Table 1. Mesh convergence analysis based on maximum Von Mises stress values

Type Mesh	Mesh Sizing (mm)	Number of Elements	Number of Nodes	Von Mises Stress Abutment, σ' (MPa)	Von Mises Stress Screw, σ' (MPa)	Percentage Variation of Abutment (%)	Percentage Variation of Screw (%)
Quadratic Tetrahedral	0.68	137654	233943	197.86	135.95	-	-
Quadratic Tetrahedral	0.67	141830	241263	201.42	136.61	1.80%	0.49%
Quadratic Tetrahedral	0.66	144294	245877	197.32	141.15	-2.04%	3.32%

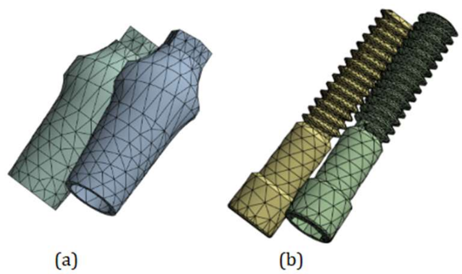

Figure 3. Meshing of the Abutment and screw

Based on literature references, the bone material was modeled as orthotropic to represent the different elastic properties in three principal directions in accordance with the anisotropic characteristics of maxillary bone. Meanwhile, the metal components, such as the fixture, abutment, and screw, were assumed to be homogeneous, isotropic, and linear elastic materials, following the standard mechanical properties of titanium alloys in the numerical simulations. The complete values of the material properties used are presented in Table 1 and serve as the basis for subsequent numerical modeling.

After all material properties were defined, numerical simulations were performed using a static structural analysis model with the boundary conditions shown in Figure 3. Fixed supports were applied to the cut side of the bone to represent the anatomical boundaries that do not move during loading. A force of 204.1 N was applied to each tooth individually, representing the average maximum bite force value in the anterior maxillary region under physiological conditions based on literature data (Kinra et al., 2017). The force direction is not perpendicular but forms a 45-degree angle to the occlusal plane, to reflect the functional force orientation during the chewing process and anterior tooth protrusion (Sakar et al., 2024). All connections between components, including fixture to abutment and abutment to screw, are modeled as bonded contacts to represent perfect osseointegration. The maxillary bone is modeled as an orthotropic material with different elastic properties in the buccolingual (E1), inferosuperior (E2), and mesiodistal (E3) directions, to represent the anisotropic behavior of bone realistically. The analysis focuses on the Von Mises stress and equivalent strain that occur in the abutment and screw as critical points of load transfer

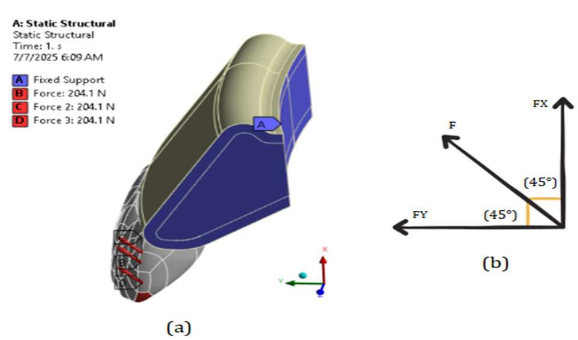

Figure 4. (a) Boundary Condition; (b) Direction of force

Table 2. Material Properties

Component of Model	Young's Modulus, E (GPa)	Shear Modulus, G (GPa)	Poisson's Ratio, ν	Density (g/cc)	References
Cortical	$E_x = 13.3$	$G_{xy} = 3.81$	$\nu_{xy} = 0.381$	-	
	$E_y = 10.8$	$G_{yz} = 4.12$	$\nu_{yz} = 0.349$	-	(C. Wang et al., 2015)
	$E_z = 19.4$	$G_{xz} = 4.63$	$\nu_{xz} = 0.224$	-	
Cancellous	$E_x = 1$	$G_{xy} = 1.15$	$\nu_{xy} = 0.3$	-	
	$E_y = 3$	$G_{yz} = 0.077$	$\nu_{yz} = 0.3$	-	(C. Wang et al., 2015)
	$E_z = 2$	$G_{xz} = 0.38$	$\nu_{xz} = 0.3$	-	
Implant Complex	113.8	-	0.342	4.43	(ASM Aerospace Specification Metals, 2025)

RESULTS AND DISCUSSION

Numerical simulation using the Finite Element Analysis (FEA) method shows that variations in implant diameter significantly affect the distribution of von Mises stress and strain, especially in the abutment and screw components. In this study, three variations of implant diameter were analyzed, namely 3.75 mm, 4.3 mm, and 5.0 mm. The stress simulation results on Abutment 1 showed an increase in the maximum value from 196.06 MPa at a diameter of 3.75 mm to 213.62 MPa at a diameter of 4.3 mm, an increase of 8.96%. However, when the diameter was enlarged to 5.0 mm, the maximum stress value decreased to 192.50 MPa, or a decrease of 9.89% compared to before. Meanwhile, on Abutment 2, the stress showed a more stable trend, decreasing from 201.42 MPa to 195.95 MPa (2.71%) and then to 191.11 MPa (2.47%). The maximum stress distribution in abutments 1 (right side) and 2 (left side), as shown in Figures 7a, 7b, and 7c, occurs at the anterior gingival portion of the abutment. This finding indicates that implant diameters that are too small or too large tend to reduce stress distribution efficiency. In contrast, intermediate diameters can trigger stress concentration due to the local stiffness of the structure under load.

In contrast, simulation results for the screw components show a different trend. In Screw 1, the maximum stress decreased from 129.26 MPa at a diameter of 3.75 mm to 115.39 MPa at a diameter of 4.3 mm (a 10.74% decrease), then further to 110.02 MPa at a diameter of 5.0 mm (a 4.66% decrease). A similar trend was observed in Screw 2, where the stress decreased from 138.61 MPa to 123.77 MPa (10.70%) and then to 118.44 MPa (4.31%). Based on Figures 7d and 7e, the maximum stress distribution in Screw 1 and 2 occurs at the center of the thread. In contrast, Figure 7f shows the distribution at the outermost edge of the screw. This decrease in stress indicates that increasing the implant diameter can produce a more even load distribution on the screw, thereby reducing the risk of stress concentration and improving the mechanical stability of the joint.

In addition to stress, the maximum strain (von Mises strain) analysis also shows interesting variations. In Abutment 1, the strain value increased from 0.00180 at a diameter of 3.75 mm to 0.00191 at a diameter of 4.3 mm (a 6.11% increase), then decreased to 0.00178 at a diameter of 5.0 mm (a 6.81% decrease). In Abutment 2, the strain value gradually decreased from 0.00185 to 0.00181 (a 2.16% decrease) to 0.00179 (a 1.10% decrease). The maximum strain distribution in abutments 1 and 2, as seen in Figures 8a, 8b, and 8c, is located at the anterior gingival portion of the abutment. These results confirm previous findings that a medium diameter (4.3 mm) is the critical point for strain concentration due to local response to loading.

Meanwhile, the strain trend in the screws appears to be more consistently decreasing. In Screw 1, the strain decreased from 0.00114 to 0.00105 (7.89%), then to 0.00100 (4.76%). Screw 2 shows a decrease from 0.00121 to 0.00109 (9.92%), then a slight increase to 0.00112 (a 2.75% increase). The maximum strain distribution in Screw 1, as shown in Figures 8d and 8f, appears at the outermost edge of the screw, while Figure 8e shows the distribution in the center of the screw. For Screw 2, Figure 8d shows the strain distribution at the center of the screw, while Figures 8e and 8f show concentrations at the outer upper edge. Although there are slight fluctuations, the general pattern observed is a decrease in elastic deformation of the screw as the implant diameter increases, which contributes to increased joint strength and overall biomechanical robustness of the system. As shown in Figures 5 and 6, the values rise and fall based on implant diameter. Thus, increasing implant diameter effectively optimizes load transmission and reduces joint stress, so selecting the ideal diameter should consider the balance between structural strength, load distribution, and overall implant system stability.

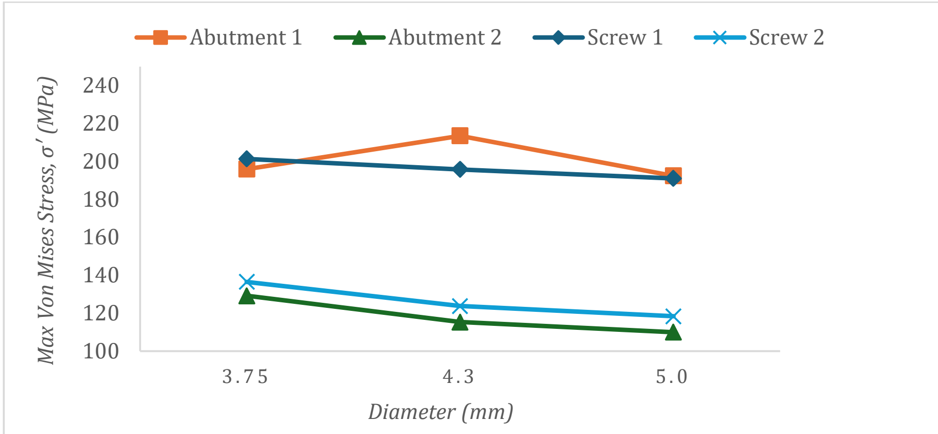

Figure 5. Von Mises stress distribution at different fixture diameters.

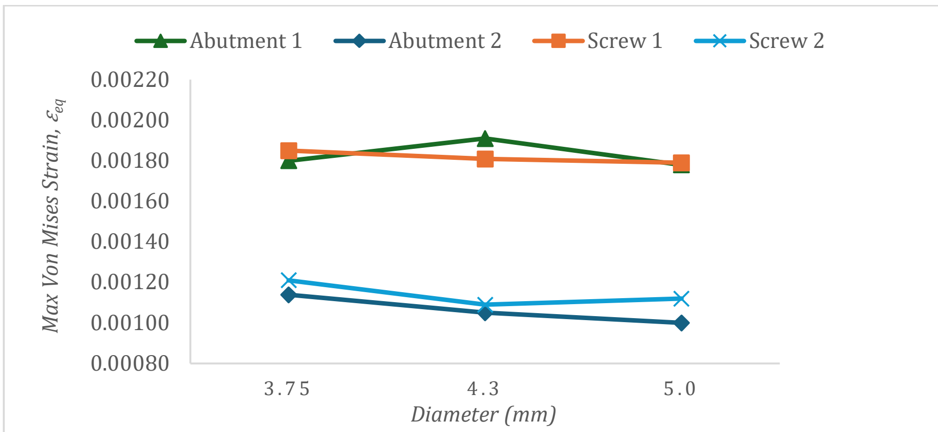

Figure 6. Von Mises strain distribution across different fixture diameters.

Table 3. FEA Results of Stress Different Fixture Diameters

Diameter (mm)	Max Von Mises Stress, σ' (MPa)			
	Abutment 1	Screw 1	Abutment 2	Screw 2
3.75	196.06	129.26	201.42	136.61
4.3	213.62	115.39	195.95	123.77
5.0	192.5	110.02	191.11	118.44

Table 4. FEA Results of Strain Different Fixture Diameters

Diameter (mm)	Max Von Mises Strain, ε_{eq}			
	Abutment 1	**Screw 1**	**Abutment 2**	**Screw 2**
3.75	0.00180	0.00185	0.00114	0.00121
4.3	0.00191	0.00181	0.00105	0.00109
5.0	0.00178	0.00179	0.00100	0.00112

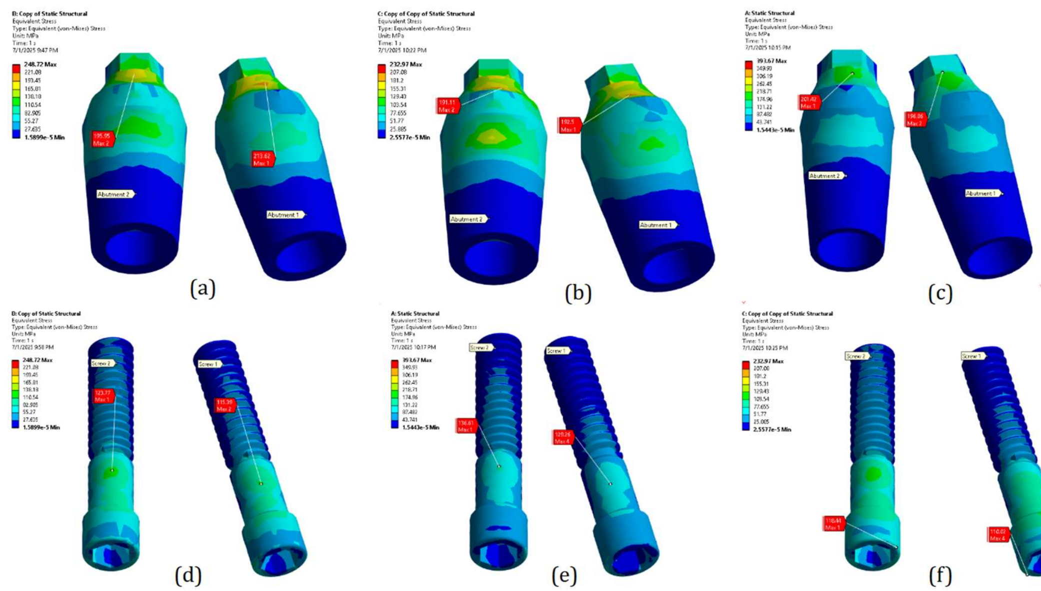

Figure 7. FEA results von mises stress: (a) Abutment at diameter 4.3 mm, (b) Abutment at diameter 5 mm, (c) Abutment at diameter 3.75 mm; (d) Screw at diameter 4.3 mm, (e) Screw at diameter 3.75 mm, (f) Screw at diameter 4 mm.

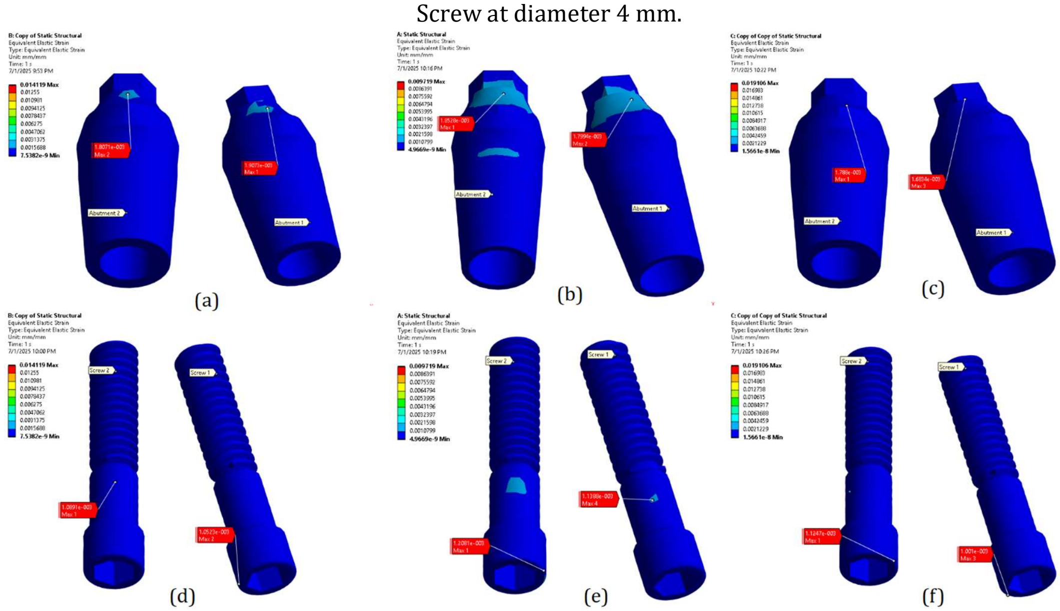

Figure 8. FEA results von mises strain: (a) Abutment at diameter 4.3 mm (b) Abutment at diameter 3.75 mm (c) Abutment at diameter 5 mm (d) Screw at diameter 4.3 mm (e) Screw at diameter 3.75 mm (f) screw at diameter 5 mm.

Table 5. Calculated FOS values for each implant component at various diameters

Diameter (mm)	Factor of Safety (FOS)			
	Abutment 1	Screw 1	Abutment 2	Screw 2
3.75	4.49	6.81	4.37	6.44
4.3	4.12	7.63	4.49	7.11
5.0	4.57	8.00	4.60	7.43

This study evaluates the factor of safety (FOS) value by comparing the simulated maximum stress with the material's elastic limit. This parameter is used to assess the structural safety margin, particularly for abutment and screw components that directly bear functional loads. In this simulation, Ti 6Al 4V material with a yield strength of 880 MPa and an elastic modulus of 113.8 GPa (ASM Aerospace Specification Metals, 2025). Table 5 is the result of calculating the value of safety factors (FOS) based on equation 4. The analysis shows that an increase in implant diameter produces a higher FOS value, especially for the screw component. This finding indicates that implants with wider diameters are able to withstand the voltage, thereby reducing the possibility of material failure or plastic deformation. This result is in line with the findings of Menacho-Mendoza et al. (2022), which analyzes elements up to (FEA) on three implant systems and concludes that FOS values greater than 1 indicate a clinically safe design. The FOS value in this study is consistently greater than 1, so the conclusion is that the bridge supported by structurally modeled implants is safe.

In addition to FOS evaluation, plastic deformation can also be estimated by comparing the strength of melting with the modulus of implant material elasticity, producing an elastic strain threshold of 0.00773. Based on the data in Table 4, the strain levels observed during the simulation vary between 0.001 and 0.00191. Basically, this study produces values below the elastic limit, which makes the Implant component fully operate. The results show the tendency that larger diameter implants produce lower strain, indicating a positive contribution to increasing biomechanical stability and decreased structural loads. The relationship between diameter and strain is increasingly emphasizing the importance of the selection of diameters in an implant installation planning process on the anterior Maxilla.

From the results carried out, several studies have done the same and produced the same results, but not specifically in the anterior Maxilla. Although limited, the conclusions or results of the existing studies can be used.. For instance, Eazhil et al. (2016) demonstrated through FEA that increasing implant diameter significantly decreased von Mises stress while enhancing bone-implant contact and overall structural stability. Jeng et al. (2020) reported similar trends in reduced stress and improved integrity as fixture diameters increased. Likewise, Alqahtani et al. (2023) It was found that larger implant diameters reduced flow rates in all components. Although this study focused on posterior or other cases, the biomechanical principles remain valid. Therefore, this study offers a valuable foundation for exploring implant diameter selection, especially in anterior regions with complex anatomical and aesthetic demands, while acknowledging the inherent limitations of FEA methodology. The assumption of using static loads in simulations does not fully represent functional conditions in the oral cavity, which involve cyclic, complex, and variable forces depending on masticatory activity. This can lead to the accumulation of material fatigue that is not detected in static models. Furthermore, while modeling bone as a homogeneous orthotropic material is more accurate than isotropic, it still fails to capture the microstructural complexity and density variations of trabecular and cortical bone in human bone.

Therefore, as a direction for future research, it is recommended to adopt multidirectional dynamic functional load-based simulations that reflect actual masticatory conditions, including the duration, frequency, and direction of loading. Furthermore, patient-specific modeling approaches based on CBCT or MRI images can construct more individualized bone geometry and mechanical properties, thus allowing simulation results to correlate more closely with actual clinical conditions. The incorporation of fatigue analysis is also crucial for evaluating the long-term durability of implant systems, particularly in anterior maxillary cases, which pose significant biomechanical and aesthetic challenges.

CONCLUSIONS

Based on the results of the discussion, the following conclusions are drawn:

1. Variations in implant diameter have been shown to have a significant effect on stress and strain distribution, where the use of a diameter of 5.0 mm can reduce the maximum stress by up to 23.7% on the abutment and 33.9% on the screw compared to a diameter of 3.75 mm.
2. Strain also decreased significantly with increasing diameter, indicating that lower elastic deformation occurred in the large diameter implant system.
3. The factor of safety (FOS) increases with increasing diameter, reaching a maximum value of 8.00 for screws and 4.60 for abutments at a diameter of 5.0 mm, indicating a safe condition against structural failure.
4. The maximum strain value (≤0.00191) remains below the elastic strain limit of the material (0.00773), indicating that no plastic deformation occurred in all tested configurations.

5. This study shows larger diameter implants can improve biomechanical stability, minimize stress concentrations, and support more efficient load distribution in maxillary anterior restorations.

REFERENCE

Alqahtani, A. R., Desai, S. R., Patel, J. R., Alqhtani, N. R., Alqahtani, A. S., Heboyan, A., Fernandes, G. V. O., Mustafa, M., & Karobari, M. I. (2023). Investigating the impact of diameters and thread designs on the Biomechanics of short implants placed in D4 bone: a 3D finite element analysis. *BMC Oral Health*, *23*(1), 1–18. https://doi.org/10.1186/s12903-023-03370-8

Alshamri, H. A., Al Moaleem, M. M., Al-Huthaifi, B. H., Al-Labani, M. A., Naseeb, W. R. B., Daghriri, S. M., Suhail, I. M., Hamzi, W. H., Illah, M. J. A., Thubab, A. Y., Aljabali, S. A., & Alnehmi, M. M. (2023). Correlation Between Maxillary Anterior Teeth and Common Facial Measurements. *Clinical, Cosmetic and Investigational Dentistry*, *15*(November), 289–300. https://doi.org/10.2147/CCIDE.S438302

ANSYS, I. (2024). *Postprocessing*. ANSYS Help Documentation. https://ansyshelp.ansys.com/public/account/secured?returnurl=////////Views/Secured/corp/v242/en/ans_thry/thy_post12.html

ASM Aerospace Specification Metals. (2025). *Titanium Ti-6Al-4V (Grade 5), Annealed*. MatWeb, LLC. https://asm.matweb.com/search/SpecificMaterial.asp?bassnum=mtp641

Bandiaky, O. N., Lokossou, D. L., Soueidan, A., Le Bars, P., Gueye, M., Mbodj, E. B., & Le Guéhennec, L. (2022). Implant-supported removable partial dentures compared to conventional dentures: A systematic review and meta-analysis of quality of life, patient satisfaction, and biomechanical complications. *Clinical and Experimental Dental Research*, *8*(1), 294–312. https://doi.org/10.1002/cre2.521

Biagi, R., Mirelli, C., Ventimiglia, R., & Ceraulo, S. (2024). Traumatic Dental Injuries: Prevalence, First Aid, and Mouthguard Use in a Sample of Italian Kickboxing Athletes. *Dentistry Journal*, *12*(10). https://doi.org/10.3390/dj12100310

Budală, D. G., Lupu, C. I., Vasluianu, R. I., Ioanid, N., Butnaru, O. M., & Baciu, E. R. (2023). A Contemporary Review of Clinical Factors Involved in Speech-Perspectives from a Prosthodontist Point of View. *Medicina (Lithuania)*, *59*(7), 1–14. https://doi.org/10.3390/medicina59071322

Budynas, R. G., & Nisbett, J. K. (2015). *Shigley's Mecha nical Engineering Design Tenth Edition* (tenth edit). McGraw-Hill Education.

Byun, S. H., Seo, J. H., Cho, R. Y., Yi, S. M., Kim, L. K., Han, H. S., On, S. W., Kim, W. H., An, H. W., & Yang, B. E. (2022). Finite Element Analysis of a New Non-Engaging Abutment System for Three-Unit Implant-Supported Fixed Dental Prostheses. *Bioengineering*, *9*(10). https://doi.org/10.3390/bioengineering9100483

CARL E. MISCH. (2008). *CONTEMPORARYIMPLANTDENTISTRY* (Third Edit). Mosby, Inc., an affiliate of Elsevier Inc.

Daryl L. Logan. (2017). *A First Course in the Finite Element Method* (Sixth Edit). Cengage Learning.

Eazhil, R., Swaminathan, S., Gunaseelan, M., Kannan, G., & Alagesan, C. (2016). Impact of implant diameter and length on stress distribution in osseointegrated implants: A 3D FEA study. *Journal of International Society of Preventive and Community Dentistry*, *6*(6), 590–596. https://doi.org/10.4103/2231-0762.195518

Falcinelli, C., Valente, F., Vasta, M., & Traini, T. (2023). Finite element analysis in implant dentistry: State of the art and future directions. *Dental Materials*, *39*(6), 539–556. https://doi.org/10.1016/j.dental.2023.04.002

Fekonja, A. (2022). Morphological Diversity of Permanent Maxillary Lateral Incisors and Their Impact on Aesthetics and Function in Orthodontically Treated Patients. *Diagnostics*, *12*(11). https://doi.org/10.3390/diagnostics12112759

González-Mederos, P., Rodríguez-Guerra, J., González, J. E., Picardo, A., & Torres, Y. (2025). A Finite Element Analysis of a New Dental Implant Design: The Influence of the Diameter, Length, and Material of an Implant on Its Biomechanical Behavior. *Materials*, *18*(12), 1–20. https://doi.org/10.3390/ma18122692

Huang, Y. C., Huang, Y. C., & Ding, S. J. (2023). Primary stability of implant placement and loading related to dental implant materials and designs: A literature review. *Journal of Dental Sciences*, *18*(4), 1467–1476. https://doi.org/10.1016/j.jds.2023.06.010

Jeng, M. D., Lin, Y. S., & Lin, C. L. (2020). Biomechanical evaluation of the effects of implant neckwall thickness and abutment screw size: A 3D nonlinear finite element analysis. *Applied Sciences (Switzerland)*, *10*(10), 10–16.

https://doi.org/10.3390/app10103471

Karthiga Devi G, Ashok V, & Pradeep D. (2022). a Retrospective Study on Prevalence of Missing Maxillary Anteriors Among Young Adult. *Journal of Pharmaceutical Negative Results*, *13*(SO4), 1345–1352. https://doi.org/10.47750/pnr.2022.13.s04.160

Kinra, P. K., Jayakumar, K., Soumithran, C. S., Soumithran, C. S., Passi, D., & Singh, M. (2017). Comparative evaluation of bite force analytical study following mandibular osteosysthesis using three-dimensional and conventional locking miniplates. *National Journal of Maxillofacial Surgery*, *10*(1), 3–7. https://doi.org/10.4103/njms.NJMS

Kriswanto, Jamari, Bayuseno, A. P., Setiawan, F., Fitriyana, D. F., Huda, K., Widodo, R. D., Dhiaulhaque, & Al Afkar, N. B. (2025). Stress and Displacement Simulation of the Posterior Dental Implants on Variations of Applied Materials and Temperatures. *Journal of Advanced Research in Applied Mechanics*, *135*(1), 164–179. https://doi.org/10.37934/aram.135.1.164179

Lee, K. Y., Shin, K. S., Jung, J. H., Cho, H. W., Kwon, K. H., & Kim, Y. L. (2020). Clinical study on screw loosening in dental implant prostheses: A 6-year retrospective study. *Journal of the Korean Association of Oral and Maxillofacial Surgeons*, *46*(2), 133–142. https://doi.org/10.5125/JKAOMS.2020.46.2.133

Losenická, J., Gajdoš, O., & Kamenský, V. (2021). Cost-utility analysis of an implant treatment in dentistry. *BMC Oral Health*, *21*(1), 1–11. https://doi.org/10.1186/s12903-021-01790-y

Menacho-Mendoza, E., Cedamanos-Cuenca, R., & Díaz-Suyo, A. (2022). Stress analysis and factor of safety in three dental implant systems by finite element analysis. *Saudi Dental Journal*, *34*(7), 579–584. https://doi.org/10.1016/j.sdentj.2022.08.006

Nicholson, J. W. (2020). Titanium Alloys for Dental Implants: A Review. *Prosthesis*, *2*(2), 100–116. https://doi.org/10.3390/prosthesis2020011

OJRICK. (2025). *OJRICK's 3D Models – CGTrader*. CGTrader. https://www.cgtrader.com/free-3d-print-models/various/various-other/ojalex-jim-exocad-library

Sakar, D., Guncu, M. B., Arikan, H., Muhtarogullari, M., Aktas, G., Reiss, N., & Turkyilmaz, I. (2024). Effect of different implant locations and abutment types on stress and strain distribution under non-axial loading: A 3-dimensional finite element analysis. *Journal of Dental Sciences*, *19*(1), 607–613. https://doi.org/10.1016/j.jds.2023.11.001

Sayed, A. J., Mohsin, S. F., Garg, K. K., Agwan, M. A. S., Tareen, S. U. K., Alruthea, M. S., & Mohan, S. (2025). Finite Element Analysis (FEA) for Influence of Variation in Dental Implant Dimensions (Length and Diameter) on Peri-implant Bone Stress/Strain Distribution: A Systematic Review. *Pakistan Journal of Medical Sciences*, *41*(1), 318–330. https://doi.org/10.12669/pjms.41.1.8991

Szajek, K., & Wierszycki, M. (2023). Screw preload loss under occlusal load as a predictor of loosening risk in varying dental implant designs. *Journal of the Mechanical Behavior of Biomedical Materials*, *148*(July), 106165. https://doi.org/10.1016/j.jmbbm.2023.106165

Takahashi, T., Kihara, M., Oki, K., Matsuzaki, T., Ayukawa, Y., Matsushita, Y., & Koyano, K. (2022). Prognosis of Implants with Implant-Supported Fixed Dental Prostheses in the Elderly Population: A Retrospective Study with a 5-to 10-Year Follow-Up. *Healthcare (Switzerland)*, *10*(7). https://doi.org/10.3390/healthcare10071250

Wang, C., Fu, G., & Deng, F. (2015). Difference of natural teeth and implant-supported restoration: A comparison of bone remodeling simulations. *Journal of Dental Sciences*, *10*(2), 190–200. https://doi.org/10.1016/j.jds.2014.11.001

Wang, Y., Zhuo, L., Yang, S., Dong, C., & Shu, P. (2025). Burden of periodontal diseases in young adults. *Scientific Reports*, *15*(1), 1–9. https://doi.org/10.1038/s41598-025-88249-0

.