Jurnal Pendidikan Teknik Mesin Vol 26, No 01, June 2026 (11-21)
http://journal.unnes.ac.id

Biomechanical Effects of Fixture Diameter on Posterior Dental Implant Abutment and Screw: A 3D FEA Study

Rachael Andika[1], Kriswanto[1], Khoirul Huda[1], Rahmat Doni Widodo[1], J. Jamari[2], Athanasius Priharyoto Bayuseno[2]

[1] Mechanical Engineering, Faculty of Engineering, Universitas Negeri Semarang, Semarang, Central Java, Indonesia.
[2] Mechanical Engineering, Faculty of Engineering, Universitas Diponegoro, Semarang, Central Java, Indonesia.
Email: andikarachaelan@students.unnes.ac.id

ABSTRACT

This study evaluates the biomechanical effects of fixture diameter on posterior dental implant stability by simulating a 3D mandibular molar model in ANSYS Workbench under a static occlusal load of 177.38 N, resulting from combined X, Y, and Z force components, to assess von Mises stress, strain, and factor of safety (FOS) in abutment and screw components for fixture diameters of 3.75, 4.0, and 5.0 mm. Results revealed that increasing the diameter to 5.0 mm reduced von Mises stress by 23.7% in the abutment and 33.9% in the screw, with strain reductions of 20% and 30%, respectively, compared to the 3.75 mm diameter. These reductions enhance mechanical stability and lower the risk of implant failure in posterior applications. The highest FOS at 5.0 mm (37.48 for abutment, 25.09 for screw) confirms maximum safety, especially for posterior mandibular applications. However, the static loading assumption limits the simulation of cyclic masticatory forces. Future studies should explore dynamic loading and in vivo validation.

Keyword: Posterior dental implant, fixture diameter, finite element analysis, von Mises stress, abutment screw

INTRODUCTION

Dental implants are a reliable method of tooth loss rehabilitation, where fixtures in the jawbone do not have adverse consequences and can transmit stresses straight to the supporting bone upon osseointegration, devoid of any soft tissue interposed between the implant and the supporting bone (Branemark, 1983). Additionally, the survival rate of dental implants shows a 10-year survival rate of 99.4% in healthy individuals and 95.9% in patients with systemic conditions (Sodnom-Ish et al., 2023). However, higher occlusal forces act on the posterior regions compared to other areas, which can contribute to an increased incidence of screw loosening, particularly in the molar region, where rates reach up to 8.5% compared to other implant sites (K.-Y. Lee et al., 2020; Turkistani et al., 2023). The choice of fixture diameter plays a critical role, especially in posterior areas where occlusal loads are higher. Since implant survival rates can vary depending on clinical conditions, it becomes important to identify the factors that support long-term success. One of these key factors is primary stability, which depends mainly on the implant's design and the quality of the surrounding bone.

The primary stability of dental implants is of paramount importance to their survival. Various factors contribute to this primary stability, including the quality of the bone and fixture design and surgical technique (Ajeebi & Alquraishi, 2020). Heimes et al. (2023) further explain that macrogeometric design features such as surface characteristics, size, and shape play a crucial role in primary stability. However, optimization studies conducted by Niroomand and Arabbeiki (2020) and Didier et al. (2020) indicate that fixture diameter is the most significant factor in reducing von Mises stress on bone. Initial stability depends on fixture design, but the abutment and other components also manage load transmission and ensure the implant functions well over time.

To complement the understanding of fixture-related biomechanics, it is important to examine the role of the abutment. The abutment serves to connect the implant with the prosthetic crown, provide support and distribute chewing forces, determine the aesthetics of the restoration, allow for adaptation to the patient's anatomy, and ensure the stability and long-term success of implant treatment (Perwez et al., 2024). However, fixture design, such as diameter, also influences stress distribution on the abutment. The use of wide-diameter fixtures increases stress on the abutment by up to 12.7% when subjected to oblique loads, although it reduces stress on the implant and cancellous bone (Moreno et al., 2023). Therefore, it is important to understand the relationship between the abutment and fixture design. While the abutment and fixture design contribute to implant stability, suboptimal combinations can increase the risk of screw loosening, a leading cause of implant failure.

Given that mechanical load transmission through the abutment affects overall implant complex, complications such as screw loosening may arise. Screw loosening can occur in one part of the implant complex, namely the abutment screw, where one of its roles is as a fixture stabilizer. Seddigh (2019) noted that screw loosening is the primary cause of dental implant failure. Screw loosening typically occurs within the first six months after loading and is most frequently observed in the molar region, particularly in implants with diameters ≥5 mm. These findings suggest that fixture design factors, including placement location and diameter size, significantly influence the risk of screw loosening (K.-Y. Lee et al., 2020). This clinical case highlights the importance of selecting the optimal fixture diameter.

The long-term reliability of implant systems is influenced not only by their mechanical design but also by the material properties of each component. Material selection is one of the key factors, as implant materials must ensure long-term stability in the oral biological environment, minimize immune responses, address degradation or biocompatibility issues, and the commonly used materials are metal alloys (Daniel S. et al., 2024). Commercially pure titanium (cpTi) and Ti-6Al-4V have gained widespread application due to their favorable bioactivity and ability to bond effectively with bone tissue. The survival rate of titanium alloys is at least 89%, and typically 97-99% can last for more than 10 years (Nicholson, 2020). Although selecting appropriate materials ensures biocompatibility, evaluating mechanical stability requires testing under simulated clinical conditions.

Various researchers have performed in vivo and in vitro experiments to evaluate the stability of dental implant fixtures. However, there is now a non-invasive approach often used to assess the biomechanical reaction between dental implants and peri-implant bone, namely finite element analysis (FEA). FEA is a numerical approach to simulate stress in complex structures by dividing the domain into small interconnected elements at points called nodes, then mathematical equations are formulated to connect the elements, resulting in a solution consistent with boundary conditions (Jawad, 2017). Researchers in dentistry frequently use FEA because of its advantages, such as the physical properties of materials remaining unchanged under applied forces and the ability to conduct repeated testing. However, errors can occur during model design and result presentation, so researchers must possess clinical insight and conduct in vitro or in vivo testing to minimize errors in their research (Vaidyanathan & Banu, 2022).

Among the various outputs derived from FEA, the factor of safety (FOS) is particularly valuable for predicting the likelihood of mechanical failure. FOS is the ratio between the allowable load and the load that can cause functional failure, thereby estimating the strength of a structure with the possibility of functional failure (Budynas & Nisbett, 2015). FOS research can be conducted in the biomedical field, such as dental implants, as in the study by Cantó-Navés et al. (2021), which also identified FOS in stress distribution analysis on single-implant-supported prostheses using 3D FEA to assess the impact of various rehabilitation materials on stress in the crown, abutment, and cortical bone under dynamic impact loads. Kriswanto et al. (2025) carried out a comparable study, which also identified the influence of variations in fixture, abutment, crown materials, and thermal loads on stress, displacement, and FOS. Additionally, research by Menacho-Mendoza et al. (2022) also used FEA to compare stress distribution and FOS in three dental implant systems with internal hexagonal connections and platform switching.

Building on the discussion of FOS, several studies have also investigated how variations in fixture diameter influence stress and strain across fixture components. Shen et al. (2024) investigated the effect of bone quality, implant diameter, and abutment screw diameter on stress distribution in fixtures, abutment screws, and bone using 3D FEA, finding that wider fixtures (5 mm) resulted in better stress distribution compared to standard diameters (4 mm). A similar FEA study by Moreno et al. (2023) on the effect of fixture diameter on stress in the fixture, abutment, and bone found that extra-short dental fixtures (5 mm) with a wide diameter (6 mm) produced better biomechanical behavior for the fixture, but have a potential risk of overloading the cortical bone and bone loss over time, especially under oblique loads. Lee et al. (2022) do some research on the influence of implant diameter, implant-abutment connection type, and cancellous bone density on the biomechanical stability of implant components and surrounding bone tissue, where fixtures with bone-level connections and small diameters resulted in higher stress concentrations on the fixture and abutment, while low-density bone showed greater maximum and minimum strains, and bone volume in the atrophy zone increased with increasing implant diameter.

Despite the increasing number of FEA studies in dental implant biomechanics, research specifically investigating the influence of fixture diameter on von Mises stress, von Mises strain, and factor of safety (FOS) in abutment and screw-abutment remains limited. This research fills the gap in current knowledge by focusing on the biomechanical implications of fixture diameter variation on abutments and screw-abutments, which previous studies have not fully explored. The novelty lies in using three-dimensional finite element analysis to simulate the mechanical behavior of posterior dental implant assemblies with varying fixture diameters of 3.75 mm, 4.0 mm, and 5.0 mm under realistic loading conditions. This study aims to quantify the influence of fixture diameter (3.75, 4.0, and 5.0 mm) on von Mises stress, von Mises strain, and factor of safety (FOS) in posterior dental implant abutments and abutment screw using three-dimensional finite element analysis (3D FEA).

METHODS

This study simplified the jawbone's 3D model into a bone block, separating it into cortical and cancellous regions. The cancellous bone has cortical dimensions of 20 mm x 15 mm x 15 mm, while the cortical bone has dimensions of 15 mm x 20 mm x 1.22 mm (Alqahtani et al., 2023; Ko et al., 2017). This study uses the bone in the first mandibular molar region as the implant placement site. The implant complex consists of a fixture, screw abutment, and abutment (Figure 1).

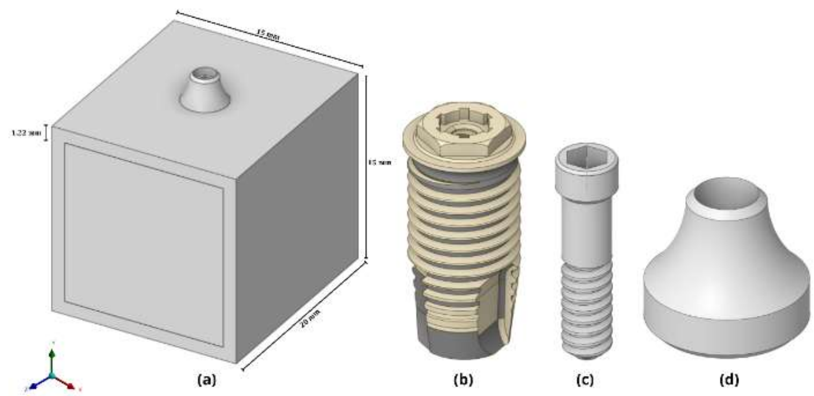

Figure 1. Components of the dental implant fe model: (a) full bone-implant assembly; (b) fixture; (c) screw between abutment and fixture; (d) abutment

The design of the implant complex components uses a commercially available design approach, namely the Nobel Biocare brand (2024), with a fixture resembling the Brånemark System MkIII TiUnite and an abutment resembling a customized snappy abutment. The fixture is a bone-level type with an external hexagon connection to the abutment. Fixture diameters have three used diameters, namely 3.75 mm, 4.0 mm, and 5.0 mm, which, according to Al-Johany et al. (2017), are considered standard diameters. Prevalence data from 13,147 implants placed between 2004 and 2012 at the Academy for Oral Implantology in Vienna, Austria, guided their selection, highlighting 3.75 mm to 4.8 mm as the most commonly used diameters (Busenlechner et al., 2014). Other dimensions remain constant, including the V-shaped thread profile, fixture length of 10 mm, thread depth of 0.3 mm, and thread pitch of 0.6 mm. To match the hexagon width listed in the catalog, the same abutment was applied for the 3.75 mm and 4.0 mm fixtures, whereas the 5.0 mm fixture has a larger abutment. This study developed the model using Autodesk Inventor Professional 2025 and performed the static simulation through ANSYS Workbench 2023 R1.

The characteristics of the material used for the implant complex material are Ti-6Al-4V material. The properties of the material model used in this study are orthotropic, homogeneous, and linearly elastic for the bone model. The implant complex is isotropic, homogeneous, and linearly elastic, as presented in Table 1.

This study applies a finite element approach based on a linear elastic model for numerical analysis. The structure's geometry was discretized into small elements, and each element contributed to the formation of an element stiffness matrix, assembled into the following global equation (Jawad, 2017):

$$[K_e][q]=[F] \quad (1)$$

Table 1. 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 = 12.7	G_{xy} = 5.5	ν_{xy} = 0.31	-	
	E_y = 22.8	G_{yz} = 7.4	ν_{yz} = 0.28	-	(Ding et al., 2014)
	E_z = 17.9	G_{xz} = 5.0	ν_{xz} = 0.18	-	
Cancellous	E_x = 0.511	G_{xy} = 0.434	ν_{xy} = 0.31	-	
	E_y = 0.907	G_{yz} = 0.081	ν_{yz} = 0.30	-	(Ding et al., 2014)
	E_z = 0.114	G_{xz} = 0.078	ν_{xz} = 0.22	-	
Implant Complex	113.8	-	0.342	4.43	(ASM Aerospace Specification Metals Inc., n.d.)

Where *[K]* is the element stiffness matrix, *[q]* is the nodal displacement vector, and *[F]* is the force vector acting on the nodal points. The element stiffness matrix $[K_e]$ is calculated based on volumetric integration:

$$[K_e] = \int_V [B]^T [D][B] \, dV \quad (2)$$

According to Hooke's law, *[B]* is the strain-displacement matrix and *[D]* is the material constitutive matrix representing the relationship between stress and strain.

Furthermore, in the FEA analysis process, one independent variable is studied in this research, namely, von Mises stress. Von Mises stress was computed using *XYZ* stress components, according to the following equation (Budynas & Nisbett, 2015):

$$\sigma' = \frac{1}{\sqrt{2}}\left[\left(\sigma_x - \sigma_y\right)^2 + \left(\sigma_y - \sigma_z\right)^2 + (\sigma_z - \sigma_x)^2 + 6\left(\tau_{xy}^2 + \tau_{yz}^2 + \tau_{zx}^2\right)\right]^{1/2} \quad (3)$$

Where σ_x, σ_y, and σ_z are the normal stresses in *XYZ* components, respectively, while τ_{xy}, τ_{yz}, and τ_{zx} are the shear stresses occurring in the *xy*, *yz*, and *zx* planes, respectively, σ' is the von Mises equivalent stress.

Another variable studied was equivalent von Mises strain, with the equation used by ANSYS being (ANSYS Inc., 2024a):

$$\varepsilon_{eq} = \frac{1}{\sqrt{2}(1+\nu)}\left[\left(\varepsilon_x - \varepsilon_y\right)^2 + \left(\varepsilon_y - \varepsilon_z\right)^2 + (\varepsilon_z - \varepsilon_x)^2 + \frac{3}{2}\left(\gamma_{xy}^2 + \gamma_{yz}^2 + \gamma_{xz}^2\right)\right]^{1/2} \quad (4)$$

Where ε_x, ε_y, and ε_z are normal strains *XYZ* components, respectively, while γ_{xy}, γ_{yz}, and γ_{xz} are shear strains occurring in the *xy*, *yz*, and *xz* planes, respectively. ε_{eq} is the von Mises equivalent strain, and ν is the effective Poisson's ratio.

The mesh to discretize the bone and implant models was a quadratic tetrahedral mesh with an element size of 0.69 mm, resolution adaptive sizing 7, and high smoothing quality. This study selected the mesh type based on its effectiveness in representing complex anatomical structures. Mesh independence was verified through a von Mises stress convergence test, using a 5% maximum allowable change as the threshold (Table 2). Additionally, mesh quality assessment uses mesh metric skewness as the evaluation standard for this study. The average skewness value obtained for the three diameter model variations is 0.4297 ± 0.2330, which is classified as "good" since it falls within the 0.25–0.5 range based on Ansys mesh quality standards (2024b). The final mesh comprised 141,632, 147,624, and 151,252 nodes for the 3.75, 4.0, and 5.0 mm diameter fixtures. Corresponding element counts were 88,900, 92,942, and 94,573. Figure 2 illustrates the mesh distribution applied in the model.

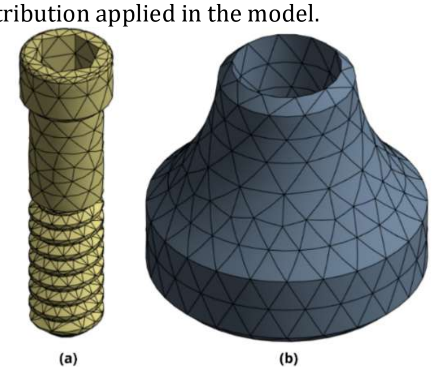

Figure 2. Meshing of the abutment and abutment screw

Force input was derived from in vivo measurements of MVC over 3 seconds on the first mandibular molar in a 62-year-old female subject. The force was applied to the outer surface of the abutment (contact area with the crown) in the form of *X*, *Y*, and *Z* vectors, with the

X-axis at -5. 8 N (toward the buccal direction), the Y-axis at -175.5 N, and the Z-axis at -25.1 N (toward the distal direction), equivalent to 177.38 N, as illustrated in Figure 3 (Bing et al., 2020).

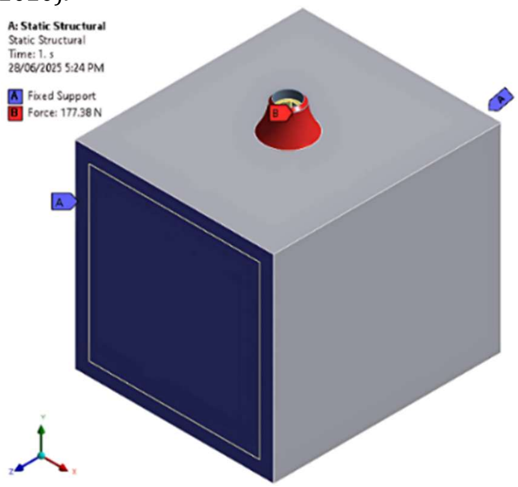

Figure 3. Boundary conditions and loading of the dental implant model

This FEA assumes 100% osseointegration, and contact between all parts is perfectly bonded. Meanwhile, fixed support is on the mesial-distal face section in the cancellous and cortical bone (Figure 3).

This study analyzes the biomechanical response in the form of Von Mises stress and Von Mises strain on the abutment and abutment screw connection due to variations in implant fixture diameter using the Finite Element Analysis (FEA) method. Additionally, the maximum Von Mises stress values are further analyzed using FOS analysis by comparing them with the yield strength of Ti-6Al-4V, which is 880 MPa (ASM Aerospace Specification Metals Inc., n.d.). The formula used is (Budynas & Nisbett, 2015):

$$n_d = \frac{S}{\sigma} \quad (5)$$

Where *nd* is FOS, *S* is yield strength, and σ is allowable stress, which in this study is maximum von Mises stress.

Table 2. Mesh convergence analysis based on maximum von mises stress values.

Type Mesh	Mesh Sizing (mm)	Number of Element	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	0.7	85664	136937	31.072	51.421	-	-
Quadratic Tetrahedral	0.69	88900	141632	30.78	53.018	-0.94%	3.11%
Quadratic Tetrahedral	0.68	90481	144208	30.885	55.468	0.34%	4.62%

RESULT & DISCUSSION

The finite element simulation results demonstrate that variations in fixture diameter correlate with changes in the maximum equivalent stress distributed across the abutment and screw components (Table 3). The fixture with a diameter of 3.75 mm generated the highest stress, measuring 30.78 MPa on the abutment and 53.011 MPa on the screw. At a diameter of 5.0 mm with a wide abutment, the stress values decreased to 23.479 MPa and 35.07 MPa, respectively, reflecting reductions of 23.7% and 33.9%. The analysis revealed that the upper edge of the abutment experienced the higher stress concentration (Figure 4). In Figure 4C, which corresponds to the largest fixture diameter combined with a wide abutment, the stress distribution appeared more favorable and evenly dispersed than Figures 4A and 4B, representing fixture diameters of 3.75 mm and 4.0 mm. This configuration reduced stress concentration at critical points on the abutment. On the screw component, the highest stress was find at the outer edge of the screw head (Figure 5). In Figure 5C, the screw also exhibited a more uniform and lower stress distribution than in Figures 5A and 5B, indicating improved mechanical behavior with the use of a larger fixture diameter. Figure 6 illustrates the stress reduction trend, with the X-axis representing fixture diameter and the Y-axis displaying maximum stress values in MPa. This figure demonstrates the inverse relationship between fixture diameter and stress magnitude.

Strain analysis revealed a similar decreasing pattern with increasing fixture diameter. The maximum strain values declined from 0.00027102 to 0.00020681 at the abutment, and from 0.00048242 to 0.00032304 at the screw, corresponding to reductions of 23.7% and 33.1%. The highest strain on the abutment was observed at the top edge, mirroring the area of stress concentration (Figure 4), while on the screw it occurred at the outer head edge (Figure 5). The relationship between fixture diameter and strain is shown graphically in Figure 7, indicating a parallel decline in both components as diameter increases.

These simulation results confirm that increasing fixture diameter consistently reduces both stress and strain on the abutment and screw. The 5.0 mm diameter fixture combined with a wide abutment achieved stress reductions of 23.7% on the abutment and 33.9% on the screw compared to the 3.75 mm diameter. This reduction pattern was also reflected in strain values, with 23.7% and 33.1% decreases on the abutment and screw, respectively. This trend reflects the effect of the enlarged cross-sectional area of the 5.0 mm abutment, which enables a wider load distribution. According to basic

Table 3. FEA results of stress and strain different fixture diameters

Diameter (mm)	Max Von Mises Stress, σ' (MPa)		Max Von Mises Strain, ε_{eq}	
	Abutment	Screw	Abutment	Screw
3.75	30.78	53.011	0.00027102	0.00048242
4.0	30.768	52.796	0.00027092	0.0004804
5.0	23.479	35.07	0.00020681	0.00032304

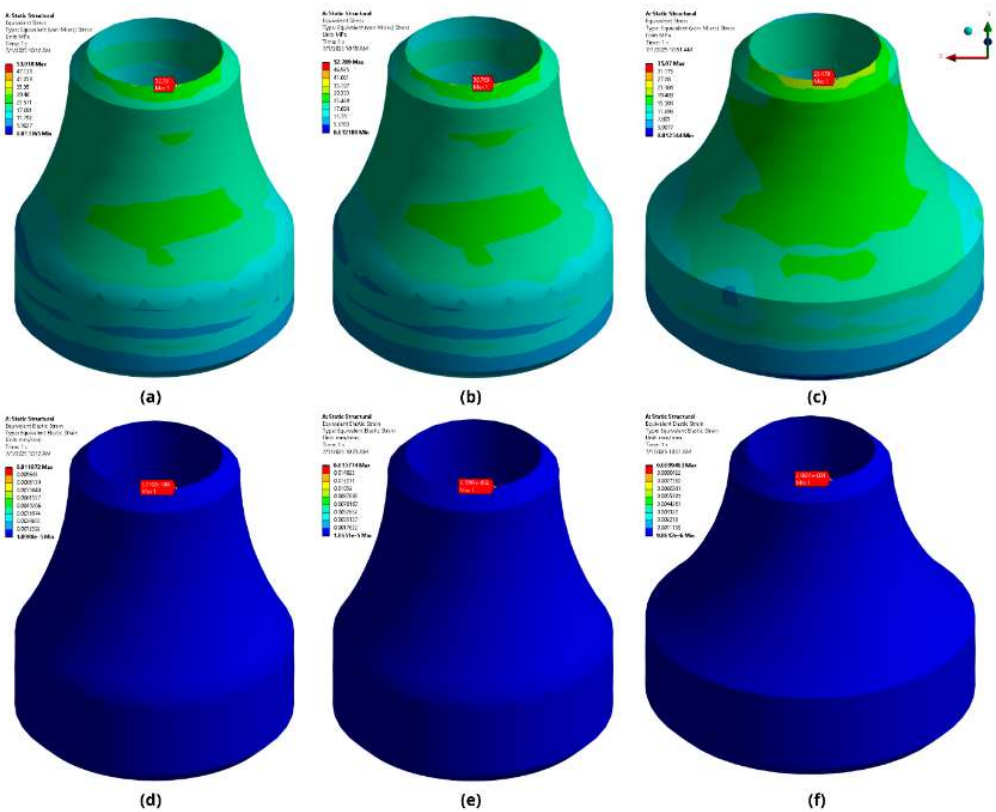

Figure 4. FEA results on abutment: (a) Von Mises stress at 3.75 mm diameter, (b) at 4.0 mm, (c) at 5.0 mm; (d) Von Mises strain at 3.75 mm, (e) at 4.0 mm, (f) at 5.0 mm.

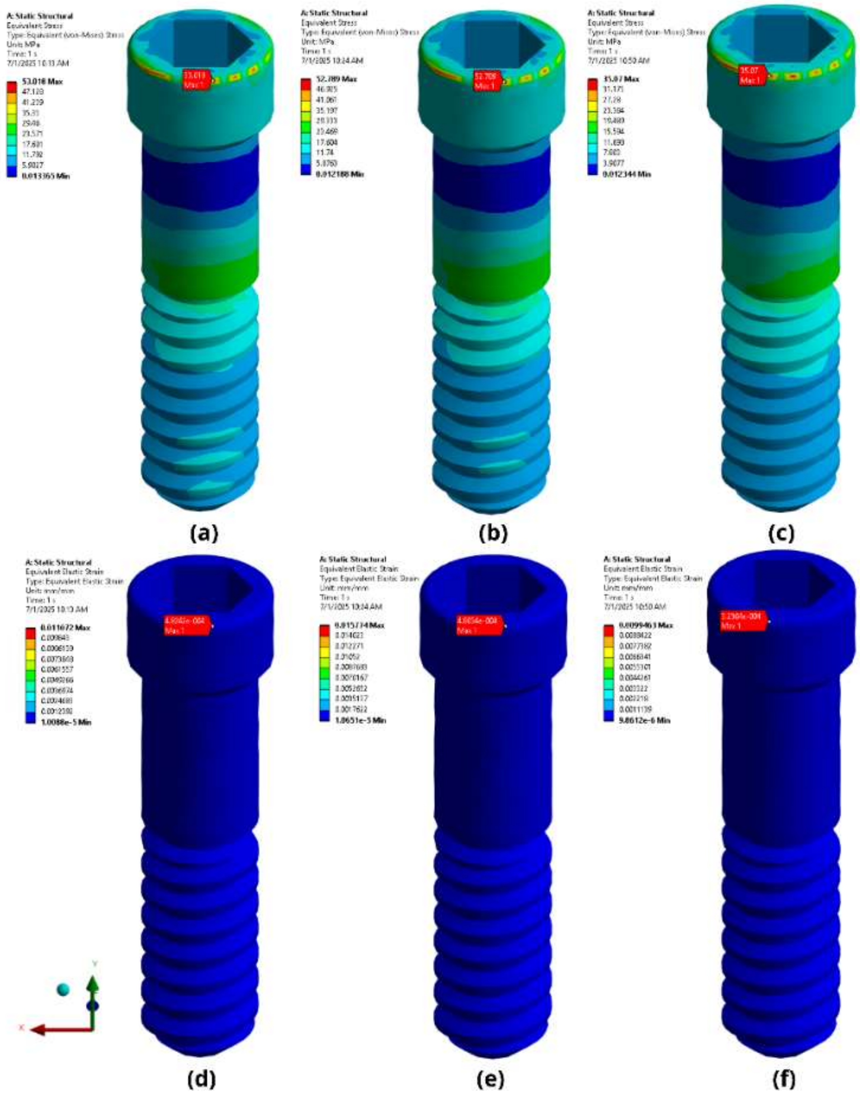

Figure 5. FEA results on abutment screw: (a) Von Mises stress at 3.75 mm diameter, (b) at 4.0 mm, (c) at 5.0 mm; (d) Von Mises strain at 3.75 mm, (e) at 4.0 mm, (f) at 5.0 mm.

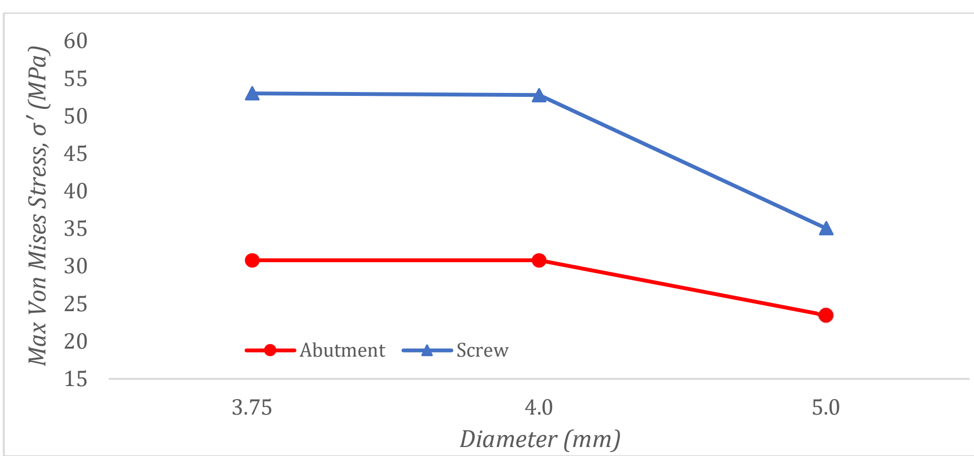

Figure 6. Graph of fixture diameter vs von mises stress

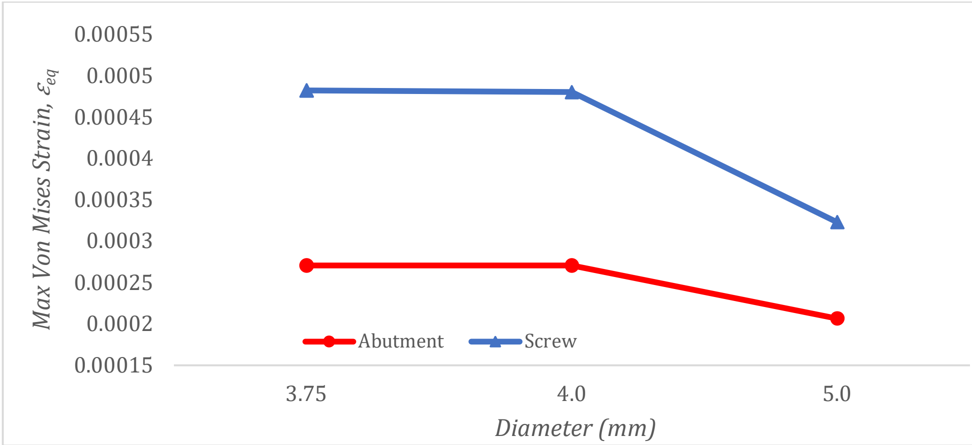

Figure 7. Graph of fixture diameter vs von mises strain

mechanical principles, stress is inversely proportional to the area *($\sigma = F/A$)*, meaning that a greater fixture diameter reduces local stress concentration. Comparatively, increasing the diameter from 3.75 mm to 4.0 mm with the exact abutment dimensions resulted in only minor decreases: 0.04% for stress on the abutment and 0.41% on the screw, and 0.04% and 0.42% reductions in strain. The slight geometric difference between these two diameters explains the marginal biomechanical improvement.

This study evaluates the fixture system's mechanical safety by comparing the obtained stress values with the material's elastic limit. In this simulation, Ti-6Al-4V was used, with a yield strength of 880 MPa and an elastic modulus of 113.8 GPa. (ASM Aerospace Specification Metals Inc., n.d.). The highest stress recorded was 53.011 MPa on the screw for the 3.75 mm fixture, and the lowest was 23.479 MPa on the abutment for the 5.0 mm fixture. These values are well below the yield threshold, indicating elastic behavior under load. The abutments' calculated factor of safety (FOS) values were 28.59, 28.60, and 37.48 for diameters of 3.75 mm, 4.0 mm, and 5.0 mm, respectively. For the screw, the FOS values were 16.60, 16.67, and 25.09. These values suggest that the 5.0 mm fixture provides the highest safety margin among the three tested diameters. Menacho-Mendoza et al. (2022) reported that an FOS value of 1 can ensure elastic deformation. In contrast, Cantó-Navés et al. (2021) recommended a more conservative threshold of ≥1.5 to mitigate clinical uncertainty. The high FOS values observed here far exceed both benchmarks, supporting the mechanical safety of the system.

Evaluation of strain also contributes to understanding the structural behavior under load. The strain values ranged between 0.00020681 and 0.00048242, valid and below the calculated elastic strain limit of 0.00773 derived from the equation $\varepsilon = \sigma/E$. These results confirm that the implant components experienced no risk of plastic deformation. Although not directly used in the FOS formula, strain indicates potential micro-movements at interfaces and deformation-induced failures. Lower strain values achieved through wider fixture diameters suggest reduced elastic elongation, improved structural stability, and enhanced resistance to microgap formation.

Several previous studies have shown consistent trends with the findings of this study. Shen et al. (2024) observed that increasing fixture and screw diameters improved stress distribution in implant assemblies. Moreno et al. (2023) emphasized the biomechanical benefit of using wider, extra-short fixtures, although they warned about potential overloading of cortical bone under oblique loading. Alqahtani et al. (2023) reported that 6 mm diameter implants produced the lowest stress, strain, and micromotion, while 4 mm implants showed the highest values. Lee et al. (2021) found similar outcomes under D4 bone conditions, reinforcing the biomechanical advantage of wider implants. Bing et al. (2020) further warned that narrow-diameter implants increase the risk of fracture due to higher stress on the fixture and abutment screw.

In addition, diameter should be prioritized during implant preparation and planning because it has the most impact on lowering von Mises stress (Niroomand & Arabbeiki, 2020; Robau-Porrua et al., 2020). Therefore, planning the use of diameter is crucial for achieving long-term stability and success of the implant. Recommendations based on diameter size indicate that a wider diameter is better regarding stress distribution.

Although the Finite Element Analysis (FEA) approach provides a systematic numerical method for simulating stress distribution in dental implant systems, this study recognizes several inherent limitations. Assuming a static load and ignoring the cyclic behavior of physiological masticatory forces may lead to underestimating fatigue failure risk during long-term function. Bone was modeled as orthotropic homogeneous, which is more accurate than isotropic assumptions, but still fails to capture the full heterogeneity of human bone tissue, potentially leading to oversimplification in stress prediction. Moreover, this study simplifies the bone geometry into a uniform bone block to facilitate computational analysis, which does not represent anatomical structures such as the mandibular arch or the varying thickness of cortical and cancellous regions, limiting load distribution accuracy in clinically relevant scenarios. The exclusion of the prosthetic crown further reduces the physiological relevance of the model, as it removes an important component that transmits occlusal force through the implant system. The model assumes a perfectly bonded condition at the implant–bone interface, omitting the possibility of micro-gaps or micromovements that could contribute to mechanical instability or bone resorption. Material fatigue behavior was not assessed, despite its importance in capturing long-term stability under repeated loading. Additionally, soft tissue structures were not included in the model, even though they contribute to force dampening and peri-implant stabilization. These limitations highlight that although FEA provides valuable biomechanical insights, the results should be interpreted carefully and complemented with in vivo or experimental validation to reflect the complexity of real clinical conditions better.

Future studies should apply dynamic loading conditions that reflect cyclic masticatory forces to better evaluate long-term fatigue risks. Implementation of CT-derived bone geometries may allow stress simulation for cortical–trabecular. Inclusion of the crown component in the model may improve the representation of occlusal load transfer. Using frictional contact modeling at the implant–bone interface may capture micro-movement potentials critical for assessing stability. Investigation into material fatigue behavior through time-dependent analysis may enhance predictions of mechanical endurance. Adding viscoelastic soft tissue layers around the implant model may reveal protective functions in force buffering and peri-implant bone maintenance. In addition, multi-variable optimization involving implant geometry, thread design, and patient-specific bone quality may provide deeper insights into factors affecting long-term implant survival.

CONCLUSION

Based on the FEA findings, this study presents the following conclusions:

1. Increasing the fixture diameter from 3.75 mm to 5.0 mm with a wide abutment reduces von Mises stress by 23.7% in the abutment and 33.9% in the screw, with stress concentrated at the abutment's upper edge and screw head's outer edge.
2. Larger fixture diameters (3.75 mm to 5.0 mm) decrease von Mises strain by 23.7% in the abutment and 33.1% in the screw, following similar distribution patterns as stress.
3. The 5.0 mm fixture diameter yields higher FOS values (37.48 for abutment, 25.09 for screw) compared to 3.75 mm (28.59 for abutment, 16.60 for screw) and 4.0 mm (28.60 for abutment, 16.67 for screw), indicating improved mechanical safety.
4. The 5.0 mm fixture diameter with a wide abutment optimizes biomechanical stability, significantly reducing stress and strain, with minimal improvements from 3.75 mm to 4.0 mm (0.04%–0.42% reductions).
5. All stress and strain values remain well below the elastic limit of Ti-6Al-4V, ensuring mechanical safety and supporting using 5.0 mm fixtures for enhanced long-term implant stability.

REFERENCE

Ajeebi, A. M., & Alquraishi, S. A. (2020). Primary stability of dental implant: a review. *International Journal of Medicine in Developing Countries*, *4*(8), 1281–1286. https://doi.org/10.24911/IJMDC.51-1586791181

Al-Johany, S. S., Al Amri, M. D., Alsaeed, S., & Alalola, B. (2017). Dental Implant Length and Diameter: A Proposed Classification Scheme. *Journal of Prosthodontics*, *26*(3), 252–260. https://doi.org/10.1111/jopr.12517

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

ANSYS Inc. (2024a). *Chapter 17: Postprocessing in ANSYS Mechanical APDL 2024 R2 Theory References*. https://ansyshelp.ansys.com/public/account/secured?returnurl=////////Views/Se cured/corp/v242/en/ans_thry/thy_post12.html

ANSYS Inc. (2024b). *Skewness in Ansys Meshing 2024 R2 User's Guide*. https://ansyshelp.ansys.com/public/account/secured?returnurl=//////Views/Secured/corp/v242/en/wb_msh/msh_skewness.html

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

Bing, L., Mito, T., Yoda, N., Sato, E., Shigemitsu, R., Han, J.-M., & Sasaki, K. (2020). Effect of peri-implant bone resorption on mechanical stress in the implant body: In vivo measured load-based finite element analysis. *Journal of Oral Rehabilitation*, *47*(12), 1566–1573. https://doi.org/10.1111/joor.13097

Branemark, P. I. (1983). Osseointegration and its experimental background. *The Journal of Prosthetic Dentistry*, *50*(3), 399–410. https://doi.org/10.1016/S0022-3913(83)80101-2

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

Busenlechner, D., Fürhauser, R., Haas, R., Watzek, G., Mailath, G., & Pommer, B. (2014). Long-term implant success at the Academy for Oral Implantology: 8-year follow-up and risk factor analysis. *Journal of Periodontal & Implant Science*, *44*(3), 102–108. https://doi.org/10.5051/jpis.2014.44.3.102

Cantó-Navés, O., Medina-Galvez, R., Marimon, X., Ferrer, M., Figueras-Álvarez, Ó., & Cabratosa-Termes, J. (2021). A 3D Finite Element Analysis Model of Single Implant-Supported Prosthesis under Dynamic Impact Loading for Evaluation of Stress in the Crown, Abutment and Cortical Bone Using Different Rehabilitation Materials. *Materials*, *14*(13). https://doi.org/10.3390/ma14133519

Daniel S., A., P., S. P. A., Naveen, J., Khan, T., & Khahro, S. H. (2024). Advancement in biomedical implant materials—a mini review. *Frontiers in Bioengineering and Biotechnology*, *Volume 12*. https://doi.org/10.3389/fbioe.2024.1400918

Didier, P., Piotrowski, B., Coz, G. Le, Joseph, D., Bravetti, P., & Laheurte, P. (2020). Finite Element Analysis of the Stress Field in Peri-Implant Bone: A Parametric Study of Influencing Parameters and Their Interactions for Multi-Objective Optimization. *Applied Sciences*, *10*(17), 5973. https://doi.org/10.3390/APP10175973

Ding, X., Liao, S.-H., Zhu, X.-H., & Wang, H.-M. (2014). Influence of Orthotropy on Biomechanics of Peri-Implant Bone in Complete Mandible Model with Full Dentition. *BioMed Research International*, *2014*(1), 709398. https://doi.org/10.1155/2014/709398

Heimes, D., Becker, P., Pabst, A., Smeets, R., Kraus, A., Hartmann, A., Sagheb, K., & Kämmerer, P. W. (2023). How does dental implant macrogeometry affect primary implant stability? A narrative review. *International Journal of Implant Dentistry*, *9*(1), 1–11. https://doi.org/10.1186/S40729-023-00485-Z

Jawad, M. H. (2017). Finite Element Analysis. In *Stress in ASME Pressure Vessels, Boilers, and Nuclear Components* (pp. 283–308). Wiley Online Library. https://doi.org/10.1002/9781119259299.ch11

Ko, Y.-C., Huang, H.-L., Shen, Y.-W., Cai, J.-Y., Fuh, L.-J., & Hsu, J.-T. (2017). Variations in crestal cortical bone thickness at dental implant sites in different regions of the jawbone. *Clinical Implant Dentistry and Related Research*, *19*(3), 440–446. https://doi.org/10.1111/cid.12468

Kriswanto, Jamari, Bayuseno, A. P., Setiawan, F., Fitriyana, D. F., Huda, K., Widodo, R. D., Dhiaulhaque, D., & 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 SE-Articles), 164–179. https://doi.org/10.37934/aram.135.1.164179

Lee, H., Jo, M., & Noh, G. (2021). Biomechanical effects of dental implant diameter, connection type, and bone density on microgap formation and fatigue failure: A finite element analysis. *Computer Methods and Programs in Biomedicine*, *200*(105863), 1–8. https://doi.org/10.1016/J.CMPB.2020.105863

Lee, H., Jo, M., Sailer, I., & Noh, G. (2022). Effects of implant diameter, implant-abutment connection type, and bone density on the biomechanical stability of implant components and bone: A finite element analysis study. *The Journal of Prosthetic Dentistry*, *128*(4), 716–728. https://doi.org/10.1016/J.PROSDENT.2020.08.042

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

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. *The Saudi Dental Journal*, *34*(7), 579–584. https://doi.org/10.1016/j.sdentj.2022.08.006

Moreno, V., Gomes, R., Ribeiro, M., Itaborai, M., Cury, A. A. Del, & Machado, R. (2023). Influence of diameter in the stress distribution of extra-short dental implants under axial and oblique load: a finite element analysis. *Brazilian Journal of Oral Sciences*, *22*(e238152), 1–14. https://doi.org/10.20396/bjos.v22i00.8668152

Nicholson, J. W. (2020). Titanium Alloys for Dental Implants: A Review. *Prosthesis*, *2*(2), 100–116. https://doi.org/10.3390/PROSTHESIS2020011

Niroomand, M. R., & Arabbeiki, M. (2020). Effect of the dimensions of implant body and thread on bone resorption and stability in trapezoidal threaded dental implants: a sensitivity analysis and optimization. *Computer Methods in Biomechanics and Biomedical Engineering*, *23*(13), 1005–1013. https://doi.org/10.1080/10255842.2020.1782390

Nobel Biocare. (2024). *Product Catalog (2024/2025 Edition)*. Nobel Biocare. https://www.nobelbiocare.com/sites/g/files/wdvifx201/files/89351Catalog2024_Aug28.pdf

Perwez, E., Sachdeva, S., & Krishna, M. (2024). Implant Abutment: A Vital Link in the Success of Implant Restoration. *International Journal of Health Sciences and Research*, *14*(5), 319–325. https://doi.org/10.52403/ijhsr.20240541

Robau-Porrua, A., Pérez-Rodríguez, Y., Soris-Rodríguez, L. M., Pérez-Acosta, O., & González, J. E. (2020). The effect of diameter, length and elastic modulus of a dental implant on stress and strain levels in peri-implant bone: A 3D finite element analysis. *Bio-Medical Materials and Engineering*, *30*(5–6), 541–558. https://doi.org/10.3233/BME-191073

Seddigh, M. A., & Mostafavi, A. S. (2019). Implant Abutment Screw Loosening: A Review of Effective Factors. *Journal of Clinical and Diagnostic Research*, *13*(8), 6–9. https://doi.org/10.7860/JCDR/2019/41751.13088

Shen, Y. W., Huang, H. L., Hsu, J. T., & Fuh, L. J. (2024). Effects of diameters of implant and abutment screw on stress distribution within dental implant and alveolar bone: A three-dimensional finite element analysis. *Journal of Dental Sciences*, *19*(2), 1126–1134. https://doi.org/10.1016/J.JDS.2023.12.020

Sodnom-Ish, B., Eo, M. Y., Kim, M. J., & Kim, S. M. (2023). A 10-year survival rate of tapered self-tapping bone-level implants from medically compromised Korean patients at a maxillofacial surgical unit. *Maxillofacial Plastic and Reconstructive Surgery*, *45*(1), 1–8. https://doi.org/10.1186/S40902-023-00401-W

Turkistani, K. A., Alkayyal, M. A., Abbassy, M. A., Al-Dharrab, A. A., Zahran, M. H., Melis, M., & Zawawi, K. H. (2023). Comparison of occlusal bite force distribution in subjects with different occlusal characteristics. *CRANIO®*, *41*(3), 204–211. https://doi.org/10.1080/08869634.2020.1830662

Vaidyanathan, A. K., & Banu, R. F. (2022). Finite element analysis – Concepts for knowledge and implementation in dental research. *The Journal of Indian Prosthodontic Society*, *22*(3). https://doi.org/10.4103/jips.jips_299_22