Biomechanical Evaluation of Implant Angulation on the First Mandibular Premolar : A 3D FEA Study

Andrean Rachman Rizaldy[1], Kriswanto[1], Septian Eko Cahyanto[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: andreanrachman@students.unnes.ac.id

ABSTRACT

The design and orientation of dental implant connections are crucial to biomechanical success, especially in the posterior mandible, which experiences severe occlusal stresses. The purpose of this study is to use a three-dimensional Finite Element Analysis (FEA) method to assess the impact of various abutment angulations (0°, 15°, and 25°) on the biomechanical behavior of dental implants. A reverse-buttress implant model was developed and tested in ANSYS Mechanical APDL 2023 R1, using a static physiological load of 240 N on an idealized mandibular bone block. The simulation assessed stress and strain distributions and calculated safety factors using Ti-6Al-4V's yield strength (880 MPa). The results revealed that increased abutment angulation greatly increased both maximum stress and strain, particularly in the screw component, where the peak stress reached 455.89 MPa at 15°. The safety factor dropped from 14.18 (0°) to 2.94 (25°), yet it remained over the acceptable safety level of 1.5. These data indicate that the 0° arrangement produces the most steady biomechanical response. Despite the simulation's power, it has limitations such as a simpler bone model and static loading. Future studies should include anatomical bone geometry and dynamic fatigue analysis to improve clinical applicability. This work emphasizes the significance of abutment angulation in implant design and endorses the use of safety factor-based FEA as a prediction technique for evaluating implant structural safety.

Keyword: Dental implant, abutment angulation, finite element analysis, safety factor, Ti–6Al–4V, mandibular premolar

INTRODUCTION

As one of the most crucial organs in the human body, teeth help with speech articulation, food eating, and face attractiveness, among other essential tasks (Ouchi & Nakagawa, 2023). Teeth are essential for preserving a person's quality of life because of these purposes (Borg-Bartolo et al., 2022). But injury to teeth can eventually result in irreversible tooth loss. Many things can lead to tooth loss or edentulism, such as dental caries, periodontal disease brought on by poor oral hygiene, or physical trauma that weakens the tooth's structure (Block et al., 2022; Gao et al., 2020). If this issue is not treated right away, it can cause a number of problems, including food buildup in empty areas, tooth movement, gum disease, imbalanced face proportions, and loss of alveolar bone (Romandini et al., 2021).

Dental implant placement is one of the most popular methods in contemporary dentistry for treating tooth loss (Kayabasi, 2020; Monaghesh et al., 2023). Dental implants are artificial devices placed into the jawbone to support prostheses like dentures, crowns, and bridges by replacing lost tooth roots (Aneksomboonpol et al., 2023; Cicciù & Tallarico, 2021). Despite its generally high success rate, implantation is not uniform throughout the oral cavity. This is because each jaw region has unique anatomical and biomechanical features (Seong et al., 2009).

The posterior mandible, particularly the first premolar area, is among the most challenging places to insert implants. This area has varied bone density (D2 to D3) and restricted vertical space due to its closeness to the mandibular canal. Furthermore, this location has very high occlusal loads, which increases the risk of stress on implant components such abutments and screws (Chaiy et al., 2011; Sugiura et al., 2015). Given these obstacles, a rigorous biomechanical approach is required to guarantee that implant placement produces optimum and safe patient outcomes.

Axial or perpendicular implants (0° angle) are often selected because they give the best vertical load distribution into the bone. However, anatomical factors frequently prevent axial implant insertion in practical reality, necessitating implantation at a certain angle, such as 15° or 20°. This tilt avoids essential anatomical features such as the mandibular canal or adapts to unequal alveolar bone loss (Hamilton et al., 2021; Mahmoudinezhad et al., 2022). Although this method is widely performed, scientific information supporting specific inclination degrees for stress and strain distribution on implant components remains sparse and varies (Oliveira et al., 2020).

Previous research has indicated that an abutment angulation of up to 25° can reduce stress on some structures; however, other studies have found that an inclination greater than 30° can increase micromotion and increase the likelihood of early failure (Atieh et al., 2023; K N et al., 2024). Furthermore, most earlier biomechanical investigations relied on basic cylindrical implant models, ignoring the anatomical complexity and unique design of components like as screws and abutments (Lopes et al., 2022). However, design features such as reverse buttresses on implants can considerably impact load distribution and stress buildup at mechanical joints.

Unlike previous studies, which generally focused solely on bone stress analysis or used generic cylindrical implant designs without considering joint design details (Atieh et al., 2023; Ibrahim et al., 2024; Oliveira et al., 2020), this study focuses on the distribution of stress and strain on mechanical joint components (abutments and screws) with variations in angle of inclination and reverse-buttress thread design that are more representative of actual clinical conditions. Furthermore, our technique incorporates yield strength-based safety factor analysis to determine the structural safety margin of each design, a topic that has received little attention in prior research. Using a thorough numerical methodology, this work provides a unique viewpoint on the biomechanical evaluation of dental implant systems, specifically for applications in the posterior mandibular area of the first premolar tooth.

Previous finite element research looked at the effects of various abutment angulations on stress distribution and micromotion. According to Oswal et al. (2024), a 15° angled abutment decreased von Mises stress more efficiently than both straight (0°) and steeper (30°) abutments in mandibular bone, demonstrating that intermediate angulations provide the best biomechanical response. Similary (Ibrahim et al., 2024) examined an All-on-Four arrangement in the mandible and discovered that angled abutments (25°) caused more deformation and stress than straight implants.

Building upon these previous findings, the present study investigates abutment angulations of 0°, 15°, and 25°, which are commonly encountered in posterior mandibular implant placement. A three-dimensional finite element approach is employed to evaluate how these abutment angulation influence the stress and strain responses in implant components, particularly the abutment and screw, under physiological occlusal loads. This setup is intended to clarify the biomechanical behavior of reverse-buttress joint designs, which have received limited attention in previous literature.

The implant utilized is a reverse-buttress design with a diameter of 5 mm, reflecting the current trend of employing large-diameter implants in the posterior area to compensate for high occlusal stresses. This work employed three-dimensional finite element analysis (FEA), a computer technique for assessing stress and strain distribution in complicated mechanical structures (Merema et al., 2021). A 3D model of the screw and abutment components was created to replicate three distinct implant angulations: 0°, 15°, and 25°. The occlusal load was delivered as a static force with a magnitude of around 240 N, based on Thomková et al. (2024) physiological chewing force distribution. This force was divided into vector components as follows: X = -13 N (buccal direction), Y = -215 N (inferior/vertical direction), and Z = -100 N (posterior direction), to accurately depict functional loading conditions in the posterior jaw.

Furthermore, in the context of implant biomechanics, stress and strain distribution must be investigated, and a quantitative method must also be used to assess the structural resistance of components to mechanical failure. One key criterion utilized in this study is the safety factor, which measures the ratio of material strength to actual working load encountered by the structure. In engineering, the safety factor is the primary indication used to determine how safe a design is from plastic deformation or fracture (Budynas & Keith Nisbett, n.d.). The application of the safety factor to implant components such as screws and abutments provides more comprehensive predictive information, particularly for assessing whether the maximum stress values that arise remain within the yield strength tolerance limits of materials like grade 5 titanium (*Titanium Technical Data - ASM Aerospace Specification Metals*, n.d.).

The safety factor value, which is based on mechanical design principles, protects against manufacturing errors, stresses, and material uncertainties. To avoid plastic deformation before reaching the maximum design load, the commonly accepted safety factor for components composed of ductile materials subjected to static loads is 1.5 to 2.0 (Budynas & Keith Nisbett, n.d.). As a result, implant components such as screws and abutments must be evaluated not only in terms of stress distribution, but also in terms of the safety margin against material failure.

This study not only assesses the distribution of Von Mises stress and strain due to variations in the angle of inclination, but also estimates the safety factor based on the yield strength of the grade 5 titanium used. With this approach, the study aims to provide predictive information regarding the safety of mechanical implant designs in various inclination configurations, as well as to support the development of evidence-based clinical guidelines in the field of implantology.

METHODS

This study is a numerical simulation based on Finite Element Analysis (FEA) that aims to analyze the effect of variations in the angle of inclination of dental implant abutments on deformation and maximum von Mises stress, as well as the safety factor in abutments and implant screws embedded in an idealized bone model in the form of a bone block. The simulation was performed using ANSYS Mechanical APDL 2023 R1 software in static structural mode.

To determine whether the maximum stress occurring in the structure is less than the yield strength of the material, Von Mises stress is used in numerical simulation data obtained from finite element software (Merema et al., 2021). As a result, this method provides a solid basis for evaluating the suitability and safety of structural designs against failure caused by loading.

Cortical bone and cancellous bone are modeled with orthotropic properties, while implant components such as fixtures, abutments, and screws use Ti-6Al-4V material (Ding et al., 2014; Titanium Technical Data - ASM Aerospace Specification Metals, n.d.). In this study, the bone model is assumed to have orthotropic, homogeneous, and linear elastic properties, while the implant complex is considered isotropic, homogeneous, and follows linear elastic behavior. All material characteristics are summarized in Table 1.

Table 1. Material Properties

	Young's Modulus (GPa)	Shear Modulus (GPa)	Poisson's Ratio	Density (g/cc)	References
Cortical bone	$E_x = 12.7$	$G_{xy} = 5.5$	$\nu_{xy} = 0.31$	-	
	$E_y = 22.8$	$G_{yz} = 7.4$	$\nu_{yz} = 0.28$	-	(Ding et al., 2014)
	$E_z = 17.9$	$G_{xz} = 5.0$	$\nu_{xz} = 0.18$	-	
Cancellous bone	$E_x = 0.511$	$G_{xy} = 0.434$	$\nu_{xy} = 0.31$	-	
	$E_y = 0.907$	$G_{yz} = 0.081$	$\nu_{yz} = 0.30$	-	(Ding et al., 2014)
	$E_z = 0.114$	$G_{xz} = 0.078$	$\nu_{xz} = 0.22$	-	
Implant model (Ti-6Al-4V)	113.8	-	0.342	4.43	(ASM Aerospace Specification Metals Inc, n.d.)

For numerical analysis, a 3D finite element approach based on a linear elastic model was used. The implant was modeled as a hollow geometry and discretized into solid elements. This approach discretizes the structure into small components, which are then sorted using the following global equation to produce an element stiffness matrix (Jawad, 2017).

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

Where $[K_e]$ represents the element stiffness matrix, $[q]$ is the nodal displacement vector, and $[F]$ is the force vector acting on each nodal point. The element stiffness matrix is derived from the volumetric integration results as follows:

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

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. This analysis focuses on von Mises equivalent stress as the main variable. This stress is calculated based on the stress components in the x, y, and z directions, as well as the shear stress on the xy, yz, and zx planes, using the following equation (Budynas & Keith Nisbett, n.d.):

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

σ_x, σ_y, and σ_z are the normal stresses in the x, y, and z directions, 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.

Additionally, von Mises equivalent strain is analyzed using the ANSYS approach (ANSYS, 2024) with the following formula:

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

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

The next step, after obtaining the Von Mises stress distribution from the simulation results, is to evaluate the safety of the implant component structure through safety factor calculations. The safety factor is an important parameter in mechanical engineering and materials engineering design. It is used to ensure that a component has sufficient margin of failure against working stress. The safety factor is defined by (Budynas & Keith Nisbett, n.d.) as the ratio between the material strength, in this case the yield strength or ultimate strength, and the maximum stress received by the component under actual working conditions. In other words, the safety factor value indicates how far the actual working load is below the maximum capacity that the material can withstand before structural failure occurs. The implant material used in this simulation is the titanium alloy Ti–6Al–4V (which is biocompatible and commonly used in dental implant applications. According to data from (*Titanium Technical Data - ASM Aerospace Specification Metals*, n.d.), the Ti–6Al–4V alloy in the annealed condition has a yield strength of 880 MPa. This value is used as a reference in determining the material's elastic limit during the finite element analysis (FEA) process. The selection of this yield strength value is important for evaluating the maximum safe stress limit based on the von Mises criterion calculated in the simulation.

Mathematically, the safety factor against plastic failure is calculated using the formula (Budynas & Keith Nisbett, n.d.):

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

Where *nd* is the safety factor, *S* is the yield strength, and the maximum working stress obtained from the simulation (in this case, the maximum Von Mises). This formula provides a linear estimate of the structural safety limit against plastic deformation. Since the material used in this study is titanium grade 5, which has ductile characteristics and an average yield strength of 880 MPa, the Von Mises stress safety factor method can be applied directly (*Titanium Technical Data - ASM Aerospace Specification Metals*, n.d.).

If the maximum stress generated from the simulation is close to or far from the *S/nd* value, the design is considered to have a high risk of plastic failure and therefore does not meet the safety criteria based on mechanical design standards. Therefore, by setting a minimum safety factor of 1.5 in accordance with the recommendation (Budynas & Keith Nisbett, n.d.), this simulation aims to ensure that all implant tilt configurations (0°, 15°, and 25°) remain within safe limits against structural failure. This approach enables predictive evaluation of the feasibility of the implant connection design based on numerical results, without the need for destructive experimental testing.

The 3D model of the jawbone is simplified into a bone block divided into cortical and cancellous bone. The cancellous bone has a cortical size of 20 mm x 15 mm x 15 mm, while the cortical bone has a size of 15 mm x 20 mm x 1.22 mm (Alqahtani et al., 2023). This bone is defined for implant placement in the first mandibular molar area. Figure 1.

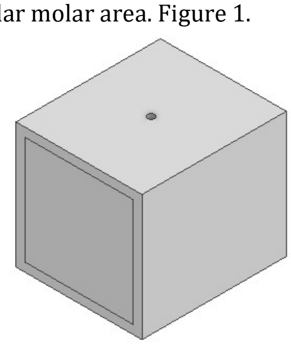

Figure 1. Bone block (Cortical bone dan Cancellous bone)

A dental implant consists of three main parts: the fixture, abutment, and screw. The design variations in this study focused on variations in the abutment angle at 0°, 15°, and 25°. The fixture design used in this study adapted the model from the Nobel Brånemark MkIII TiUnite system, but with dimensional modifications for simulation purposes. The implant model was created with variations in diameter (5 mm), total length (8.5 mm), thread pitch (0.8 mm), and thread depth (0.4 mm). The thread shape used was a reverse buttress with a taper angle of 7°, and the entire modeling process was performed using Autodesk Inventor Professional 2025 software, resulting in the design shown in Figure 2.

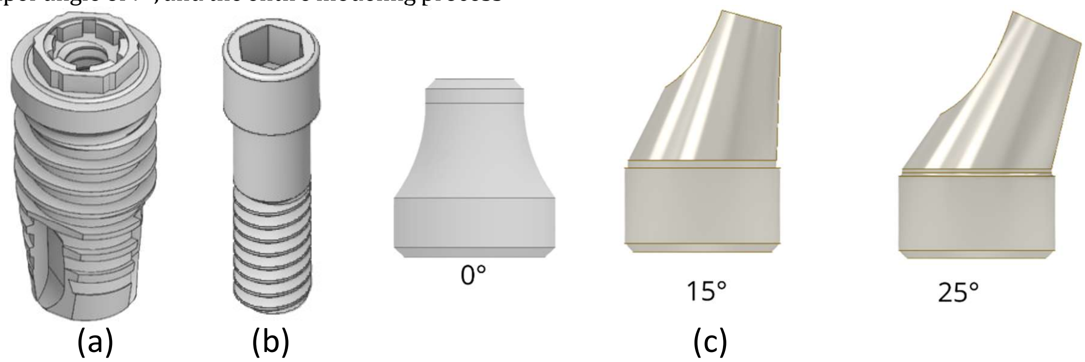

Figure 2. Implant component models: (a) fixture with reverse buttress thread design, (b) abutment screw, and (c) abutment angulation variations of 0°, 15°, and 25°.

The 3D model of the implant system was meshed using quadratic tetrahedral elements (Tetra10) in ANSYS Mechanical (Merema et al., 2021). To ensure the independence of the results from the element size, a mesh convergence test was performed with three mesh size variations: 0.68 mm, 0.67 mm, and 0.66 mm. As the element size decreased, the number of elements increased from 36,719 to 38,221, and the number of nodes increased from 64,893 to 67,447. The maximum stress distribution (von Mises) on the abutment showed a value of 148.12 MPa on the 0.68 mm mesh and increased to 151.03 MPa on the 0.66 mm mesh, resulting in a relative error of 1.49%. On the screw, the relative error is even smaller, at 0.36%, indicating that the simulation results have achieved convergence as they are below the 5% tolerance threshold(Serrato-Pedrosa et al., 2025; Thomková et al., 2024).

In addition, mesh quality was analyzed using the skewness metric, with average values ranging from 0.48894 to 0.49693 and standard deviations between 0.22892 and 0.23139, indicating a good and stable distribution of mesh elements throughout the model. The maximum skewness value is close to 1 (0.99996), but the low minimum value (<0.01) and average value still within the ideal range (<0.5) indicate that the mesh quality is still acceptable for structural analysis (Serrato-Pedrosa et al., 2025).

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

Based on these results, a mesh size of 0.67 mm was selected as the final configuration used in all simulations, as it provided an optimal balance between result accuracy, mesh quality, and computational time efficiency.

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)	Relative Error Abutment (%)	Relative Error Screw(%)
Quadratic Tetrahedral	0.68	2	36719	64893	148.12	-	-
Quadratic Tetrahedral	0.67	2	37400	65994	148.82	0.47%	3.65%
Quadratic Tetrahedral	0.66	2	38221	67447	151.03	1.49%	0.36%

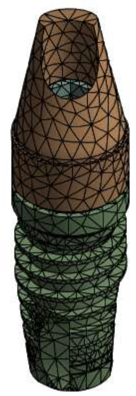

Figure 3. Meshing of the Abutment, Screw Fixture-Abutment, and Fixture

This simulation employs a global coordinate system with the following physiological orientation: the Y+ axis points superior (upward), the Z+ axis points anterior, and the X+ axis points lingually. This is consistent with the architecture of the posterior mandible, where the first premolar implant is positioned. Occlusal load is applied as a static force with a total magnitude following the physiological chewing force distribution based on the biomechanical model used by (Thomková et al., 2024). The force values are divided into vector components according to the following directions: X: −13 N (buccal direction) Y: −215 N (inferior/vertical compression direction) Z: −100 N (posterior direction) These forces are applied to the upper surface of the abutment as a surface effect to represent physiological masticatory loads angled relative to the vertical axis. Boundary conditions (support) are applied to the bone block's anterior and posterior cutting surfaces, with all degrees of freedom for translation and rotation locked (fully fixed support). This boundary setting aims to stabilize the model and prevent structural changes in the global system. It aligns with implant modeling strategies in bone as used in FEA-based biomechanics literature (Merema et al., 2021). All component contacts (bone–fixture, fixture–screw, screw–abutment) are set as bonded contacts, assuming complete osseointegration and stable implant connections without relative movement Figure 4.

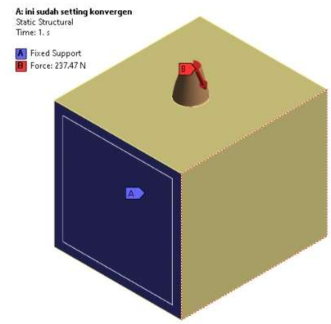

Figure 4. Boundary Conditions and Loading of the Dental Implant Model

RESULT & DISCUSSION

The effect of abutment angulation on maximum stress (von Mises stress) and maximum strain (von Mises strain) in the two main components of the implant system the abutment and screw was investigated using simulation. Three different abutment angulation variations of 0°, 15°, and 25° were evaluated under constant load conditions.

Based on the data, as the inclination angle increases, the maximum stress and strain also increase (Mosharraf et al., 2022). The maximum stress value on the abutment component increased significantly from 62.077 MPa at 0° to 298.64 MPa at 15° and 299.64 MPa at 25°. Similarly, the maximum strain shows a similar increasing pattern from 5.79×10^{-4} to 2.64×10^{-3} and remains constant at an angle of 25°.

Table 3. Inclination anFEA Results of Stress and Strain Different Abutmen Inclination

Abutmen Inclination	Max Von Mises Stress, σ' (MPa)		Max Von Mises Strain, ε_{eq}	
	Abutment	Screw	Abutment	Screw
0°	62.077	247.91	5.79E-04	2.20E-03
15°	298.64	455.89	2.64E-03	4.21E-03
25°	299.64	452.46	2.64E-03	4.29E-03

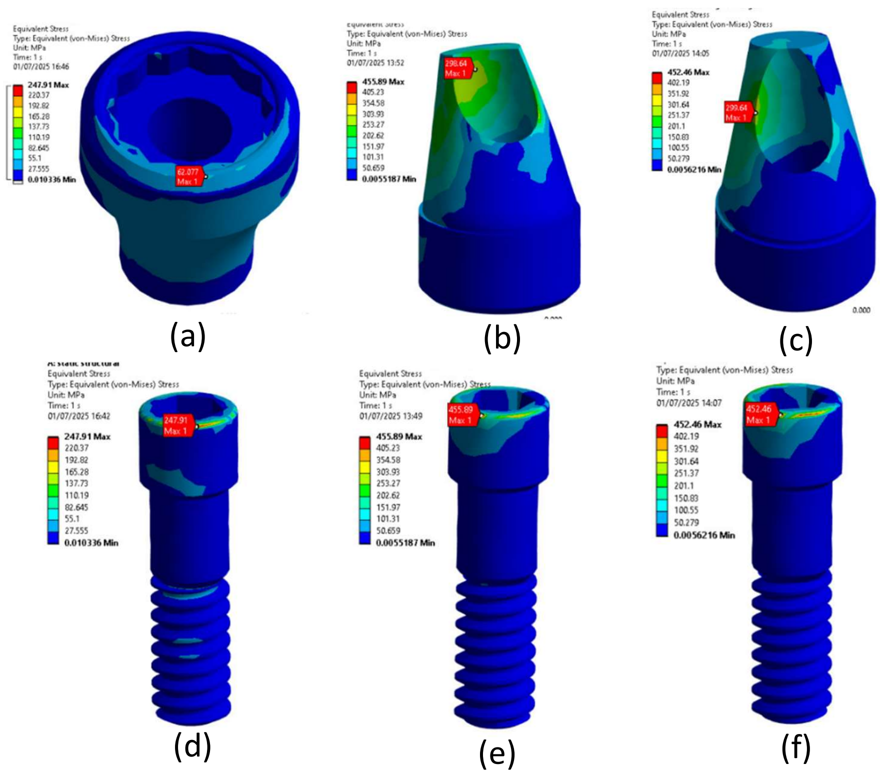

Figure 5. Von Mises stress on abutmen: (a) at 0°, (b) at 15°, (c) at 25°; Von Mises stress on screw : (d) at 0°, (e) at 15°, (f) at 25°

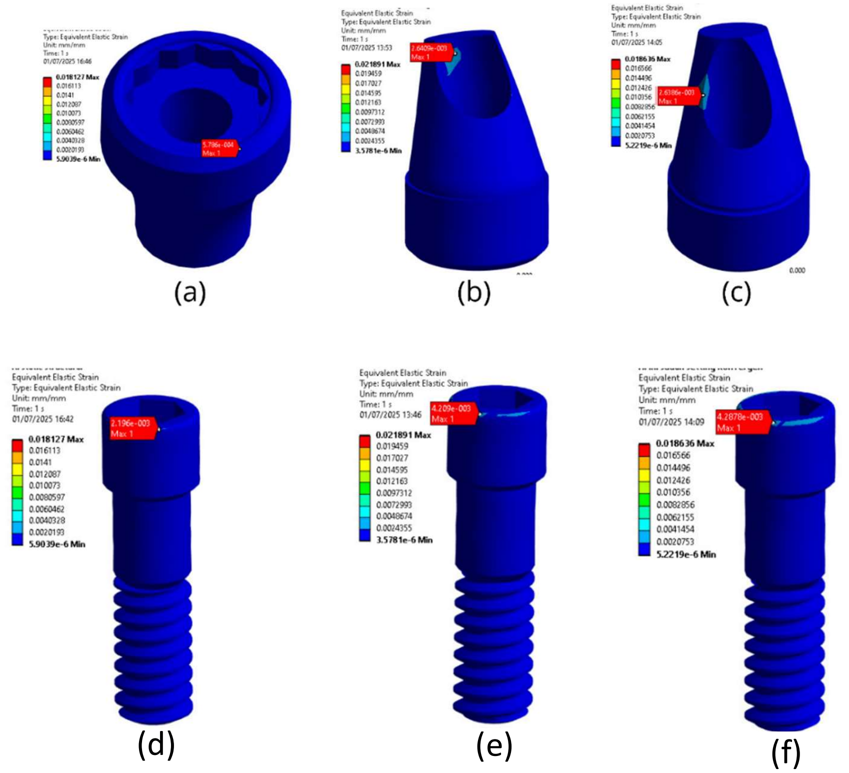

Figure 6. Von Mises strain on abutmen: (a) at 0°, (b) at 15°, (c) at 25°; Von Mises strain on screw : (d) at 0°, (e) at 15°, (f) at 25°

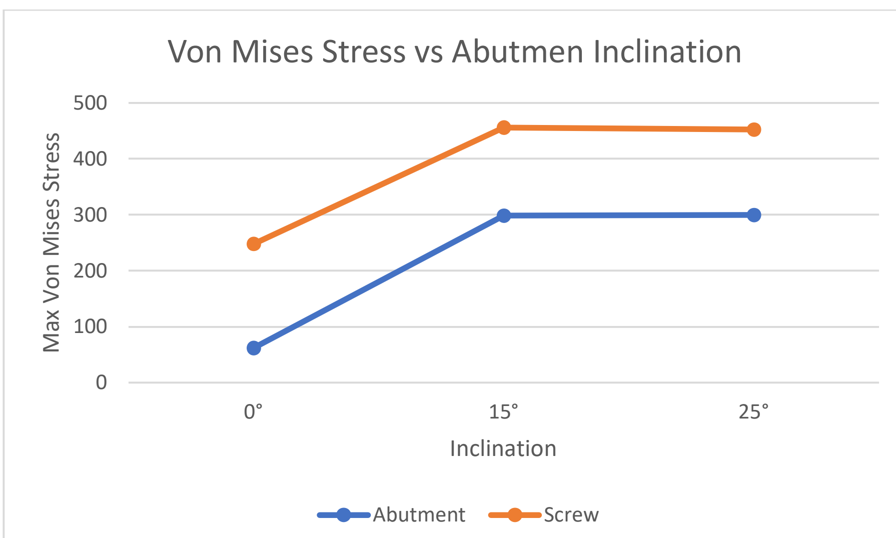

Figure 7. Maximum Von Mises stress values (MPa) on abutments and screws for abutment angles of 0°, 15°, and 25°.

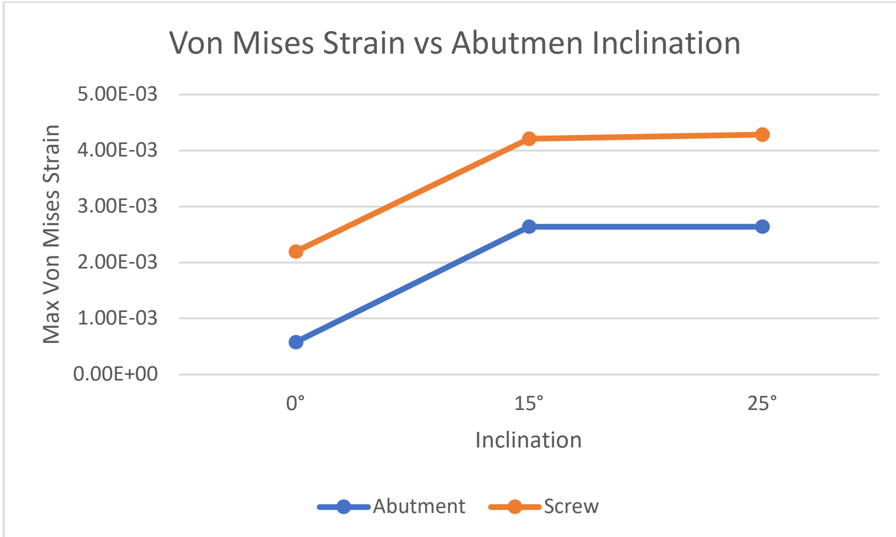

Figure 8. Maximum Von Mises strain values at the abutment and screw for abutment angles of 0°, 15°, and 25°.

Through a three-dimensional finite element method-based numerical analysis process, maximum Von Mises stress and strain data were obtained for the abutment and dental implant screw system with abutment angle variations of 0°, 15°, and 25°. The data analysis results show that at an angle of 0°, the maximum stress value on the abutment is 62.077 MPa, while on the screw it is 247.91 MPa. When the abutment angle is increased to 15°, there is a significant increase in stress, namely 298.64 MPa for the abutment and 455.89 MPa for the screw. At an angle of 25°, the maximum stress value at the abutment increased slightly to 299.64 MPa, while the screw showed a slight decrease to 452.46 MPa. In general, the highest maximum stress occurred at the screw with an abutment angle of 15°, which was 455.89 MPa.

A proportionate increasing trend was seen in maximal Von Mises stress. At a 0° angle, the highest stress on the abutment was observed at 5.79×10^{-4} and on the screw at 2.20×10^{-3}. Increasing the abutment angle to 15° increased both abutment and screw stress to 2.64×10^{-3} and 4.21×10^{-3}, respectively. When the angle was adjusted to 25°, the abutment's maximum stress remained at 2.64×10^{-3}, but the screw rose slightly to 4.29×10^{-3}. The screw had the most excellent maximum strain value measured at a 25° abutment angle.

Thus, based on the total stress and strain distribution, an abutment angulation of 0° is the most appropriate design for implant applications in the first premolar mandibular area. This angle produces the lowest stress and strain values on both the abutment and the screw, potentially lowering the danger of excessive stress concentration and preserving the implant structure. This conclusion is consistent with prior studies demonstrating that axial implant placement in the posterior jaw produces a safer and more stable biomechanical load distribution (Baggi et al., 2008)

Safety factor analysis was performed based on the maximum stress (Von Mises stress) and maximum strain generated on the abutment and screw for each variation of the angle of inclination. The yield strength value of Ti–6Al–4V material, which is 880 MPa, was used as a reference, with an elastic modulus of 113.8 GPa. The safety factor calculation for stress was performed by dividing the yield strength value by the maximum Von Mises stress, while for strain, it was performed by dividing the yield strength by the product of Von Mises strain and elastic modulus. Based on the simulation results, the stress safety factor values for the abutment were 14.18, 2.95, and 2.94 at angles of 0°, 15°, and 25°, respectively. Meanwhile, the stress safety factor values for the screw were 3.55, 1.93, and 1.94 for each angle variation. The results of the calculations based on strain show safety factor values for the abutment of 13.36, 2.93, and 2.93, while for the screws, the values are 3.52, 1.84, and 1.80. When compared to the minimum safety limit recommended by (Budynas & Keith Nisbett, n.d.), which is 1.5, all components are still within the safety limit, despite a significant decrease at the 15° and 25° angles. This indicates that increasing the abutment angle results in a significant increase in stress and strain, particularly in the screw section(Erdoğdu et al., 2024; Mosharraf et al., 2022).

The findings of this investigation show that raising the abutment angle regularly produces more stress and strain buildup in both the abutment components and the screws. Load distribution at angles of 15° and 25° causes larger stress concentrations, especially at the screw-abutment contact, which is a crucial site for mechanical failure. This trend is consistent with the findings of (Baggi et al., 2008), who discovered that abutment inclination might cause greater bending moments and lateral strains, especially in the posterior mandibular area, which bears large masticatory loads. Although the safety factor values for all abutment angulation remain above the recommended safe limit (1.5), the 0° configuration provides a significantly larger safety margin. Therefore, an abutment angulation of 0° is more mechanically recommended as it results in the lowest stress and strain levels and the highest safety factor.

Additionally, the significant reduction in the safety factor at the 15° and 25° angles warrants further attention. Although both configurations are still acceptable within safe limits according to mechanical design standards, the reduction in the safety factor to near the lower limit (1.80–1.94 for the screw) indicates that additional clinical loads, individual anatomical variations, or partial osseointegration failure could push the structure into the critical failure zone. Therefore, it is important for clinicians to carefully consider the implant inclination angle, especially in cases with suboptimal bone density such as type D3, which is commonly found in the mandibular first premolar region. Adjusting the connection geometry, using screws with larger diameters, or modifying the thread design could be alternative techniques to enhance structural resistance in inclined conditions.

Although the numerical simulation approach in this study has been able to detail the distribution of stress and strain, there are several limitations that can be improved in future research. First, the bone block model used does not fully reflect the anatomical complexity of the mandibular bone, so it is recommended to use an anatomical model based on CT or CBCT images. Second, this simulation still uses ideal static loads and does not consider dynamic loading cycles or repetitive loads (fatigue), even though actual clinical conditions are dynamic. Therefore, further studies toward multiphysics simulation or cyclic loading could provide a more realistic and predictive picture of the long-term performance of the implant system.

CONCLUSION

Based on the results of the finite element analysis, several key biomechanical insights were obtained regarding the influence of abutment angulation on the implant–abutment–screw assembly in the posterior mandibular region :

1. The abutment angle significantly affects the distribution of stress and strain on implant connection components (abutment and screw) in the posterior mandible.
2. Increasing the abutment angle from 0° to 15° and 25° causes an increase in maximum stress and strain, particularly on the screw component, which exhibits the highest stress concentration.
3. The 0° abutment inclination configuration produces the lowest stress and strain levels, as well as providing the most stable biomechanical load distribution.
4. The safety factor decreases as the abutment angle increases, but all configurations (0°, 15°, and 25°) remain within the safe range (>1.5), in accordance with recommended mechanical standards.
5. The highest safety factor was recorded at 0° abutment (14.18), while the lowest was at the 15° screw angle (1.93), indicating an increased risk of failure in inclined configurations.
6. The critical point of stress accumulation occurs at the screw–abutment joint interface, which should be a primary focus in implant joint design.
7. The reverse-buttress model with a 7° taper angle and large diameter (5 mm) proved sufficiently effective in distributing loads under axial conditions, but its effectiveness decreases under extreme inclinations.
8. Safety factor-based FEA simulations provide a robust predictive approach for assessing implant design safety, particularly in non-axial configurations common in complex clinical conditions.
9. A 0° abutment configuration is biomechanically recommended for implantation in the mandibular first premolar because it provides the greatest safety margin under static load.

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/TABLES/9

Aneksomboonpol, P., Mahardawi, B., Na Nan, P., Laoharungpisit, P., Kumchai, T., Wongsirichat, N., & Aimjirakul, N. (2023). Surface structure characteristics of dental implants and their potential changes following installation: a literature review. *Journal of the Korean Association of Oral and Maxillofacial Surgeons*, *49*(3), 114–124. https://doi.org/10.5125/JKAOMS.2023.49.3.114

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

Atieh, M. A., Shah, M., Ameen, M., Tawse-Smith, A., & Alsabeeha, N. H. M. (2023). Influence of implant restorative emergence angle and contour on peri-implant marginal bone loss: A systematic review and meta-analysis. *Clinical Implant Dentistry and Related Research*, *25*(5), 840–852. https://doi.org/10.1111/CID.13214;JOURNAL:JOURNAL:17088208;PAGE:STRING:ARTICLE/CHAPTER

Baggi, L., Cappelloni, I., Di Girolamo, M., Maceri, F., & Vairo, G. (2008). The influence of implant diameter and length on stress distribution of osseointegrated implants related to crestal bone geometry: A three-dimensional finite element analysis. *Journal of Prosthetic Dentistry*, *100*(6), 422–431. https://doi.org/10.1016/S0022-3913(08)60259-0

Block, C., König, H. H., & Hajek, A. (2022). Oral health and quality of life: findings from the Survey of Health, Ageing and Retirement in Europe. *BMC Oral Health*, *22*(1), 1–9. https://doi.org/10.1186/S12903-022-02599-Z/TABLES/3

Borg-Bartolo, R., Roccuzzo, A., Molinero-Mourelle, P., Schimmel, M., Gambetta-Tessini, K., Chaurasia, A., Koca-Ünsal, R. B., Tennert, C., Giacaman, R., & Campus, G. (2022). Global prevalence of edentulism and dental caries in middle-aged and elderly persons: A systematic review and meta-analysis. *Journal of Dentistry*, *127*, 104335. https://doi.org/10.1016/J.JDENT.2022.104335

Budynas, R. G., & Keith Nisbett, J. (n.d.). *Shigley's Mechanical Engineering Design 10th Edition*.

Chaiy, R., Pimduen, R., Qing Li, & Michael v Swain. (2011). Effects of Occlusal Inclination and Loading on Mandibular Bone Remodeling: A Finite Element Study. *Research Gacte*. https://www.researchgate.net/publication/51236888_Effects_of_Occlusal_Inclination_and_Loading_on_Mandibular_Bone_Remodeling_A_Finite_Element_Study

Cicciù, M., & Tallarico, M. (2021). Dental Implant Materials: Current State and Future Perspectives. *Materials 2021, Vol. 14, Page 371*, *14*(2), 371. https://doi.org/10.3390/MA14020371

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

Erdoğdu, M., Demirel, M. G., Mohammadi, R., & Güntekin, N. (2024). Assessment of the Impact of Bone Quality and Abutment Configuration on the Fatigue Performance of Dental Implant Systems Using Finite Element Analysis (FEA). *Journal of Personalized Medicine 2024, Vol. 14, Page 1040*, *14*(10), 1040. https://doi.org/10.3390/JPM14101040

Gao, S. S., Chu, C. H., & Young, F. Y. F. (2020). Oral Health and Care for Elderly People with Alzheimer's Disease. *International Journal of Environmental Research and Public Health 2020, Vol. 17, Page 5713*, *17*(16), 5713.

https://doi.org/10.3390/IJERPH17165713

Hamilton, A., Jamjoom, F. Z., Alnasser, M., Starr, J. R., Friedland, B., & Gallucci, G. O. (2021). Tilted versus axial implant distribution in the posterior edentulous maxilla: A CBCT analysis. *Clinical Oral Implants Research*, *32*(11), 1357–1365. https://doi.org/10.1111/CLR.13836

Ibrahim, C. R. M., Sameh, A., & Askar, O. (2024). A finite element analysis study on different angle correction designs for inclined implants in All-On-Four protocol. *BMC Oral Health*, *24*(1), 1–14. https://doi.org/10.1186/S12903-024-04091-2/FIGURES/16

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

K N, C., Eram, A., Shetty, N., Shetty, D. D., Futane, M., & Keni, L. G. (2024). Evaluating Angled Abutments: Three-Dimensional Finite Element Stress Analysis of Anterior Maxillary Implants. *Prosthesis 2024, Vol. 6, Pages 315-328*, *6*(2), 315–328. https://doi.org/10.3390/PROSTHESIS6020024

Kayabasi, O. (2020). Design methodology for dental implant using approximate solution techniques. *Journal of Stomatology, Oral and Maxillofacial Surgery*, *121*(6), 684–695. https://doi.org/10.1016/J.JORMAS.2020.01.003

Lopes, G. da R. S., Matos, J. D. M. de, Queiroz, D. A., Tribst, J. P. M., Ramos, N. de C., Rocha, M. G., Barbosa, A. B., Bottino, M. A., Borges, A. L. S., & Nishioka, R. S. (2022). Influence of Abutment Design on Biomechanical Behavior to Support a Screw-Retained 3-Unit Fixed Partial Denture. *Materials 2022, Vol. 15, Page 6235*, *15*(18), 6235. https://doi.org/10.3390/MA15186235

Mahmoudinezhad, S. S., Aryankia, A., Shooshtari, S. S., & Moradi, K. (2022). The Effect of Mandibular Angulation on Preoperative Assessment of Dental Implant Insertion at Premolar Region: CBCT Study. *BioMed Research International*, *2022*(1), 7879239. https://doi.org/10.1155/2022/7879239

Merema, B. B. J., Kraeima, J., Glas, H. H., Spijkervet, F. K. L., & Witjes, M. J. H. (2021). Patient-specific finite element models of the human mandible: Lack of consensus on current set-ups. *Oral Diseases*, *27*(1), 42–51. https://doi.org/10.1111/ODI.13381;PAGE:STRING:ARTICLE/CHAPTER

Monaghesh, E., Negahdari, R., & Samad-Soltani, T. (2023). Application of virtual reality in dental implants: a systematic review. *BMC Oral Health*, *23*(1), 1–13. https://doi.org/10.1186/S12903-023-03290-7/TABLES/2

Mosharraf, R., Abbasi, M., & Givehchian, P. (2022). The Effect of Abutment Angulation and Crown Material Compositions on Stress Distribution in 3-Unit Fixed Implant-Supported Prostheses: A Finite Element Analysis. *International Journal of Dentistry*, *2022*. https://doi.org/10.1155/2022/4451810,

Oliveira, H., Velasco, A. B., Ríos-Santos, J. V., Lasheras, F. S., Lemos, B. F., Gil, F. J., Carvalho, A., & Herrero-Climent, M. (2020). Effect of Different Implant Designs on Strain and Stress Distribution under Non-Axial Loading: A Three-Dimensional Finite Element Analysis. *International Journal of Environmental Research and Public Health 2020, Vol. 17, Page 4738*, *17*(13), 4738. https://doi.org/10.3390/IJERPH17134738

Oswal, M., Shetty, M., Hegde, C., & John, N. (2024). Effect of angulated abutments on stress concentration at the implant abutment interface: a finite element analysis. *Journal of Osseointegration*, *16*(2), 106–113. https://doi.org/10.23805/JO.2024.635

Ouchi, T., & Nakagawa, T. (2023). Cellular Signaling for Dental Physiological Functions. *Biomolecules 2023, Vol. 13, Page 1177*, *13*(8), 1177. https://doi.org/10.3390/BIOM13081177

Romandini, M., Baima, G., Antonoglou, G., Bueno, J., Figuero, E., & Sanz, M. (2021).

Periodontitis, Edentulism, and Risk of Mortality: A Systematic Review with Meta-analyses. *Journal of Dental Research*, *100*(1), 37–49. https://doi.org/10.1177/0022034520952401/SUPPL_FILE/DS_10.1177_0022034520952401.PDF

Seong, W. J., Kim, U. K., Swift, J. Q., Hodges, J. S., & Ko, C. C. (2009). Correlations between physical properties of jawbone and dental implant initial stability. *Journal of Prosthetic Dentistry*, *101*(5), 306–318. https://doi.org/10.1016/S0022-3913(09)60062-7,

Serrato-Pedrosa, J. A., Villanueva-Fierro, I., Marquet-Rivera, R. A., Hernández-Vázquez, R. A., Cruz-Lopez, S., & Loera-Castañeda, V. (2025). Non-Linear Biomechanical Evaluation and Comparison in the Assessment of Three Different Piece Dental Implant Systems for the Molar Region: A Finite Element Study. *Journal of Functional Biomaterials*, *16*(1). https://doi.org/10.3390/JFB16010017,

Sugiura, T., Yamamoto, K., Kawakami, M., Horita, S., Murakami, K., & Kirita, T. (2015). Influence of bone parameters on peri-implant bone strain distribution in the posterior mandible. *Medicina Oral, Patologia Oral y Cirugia Bucal*, *20*(1), e66–e73. https://doi.org/10.4317/MEDORAL.19878,

Thomková, B., Marcián, P., Borák, L., Joukal, M., & Wolff, J. (2024). Biomechanical performance of dental implants inserted in different mandible locations and at different angles: A finite element study. *The Journal of Prosthetic Dentistry*, *131*(1), 128.e1-128.e10. https://doi.org/10.1016/J.PROSDENT.2023.10.012

Titanium Technical Data - ASM Aerospace Specification Metals. (n.d.). Retrieved July 3, 2025, from https://www.aerospacemetals.com/titanium-technical-data/?utm_source=chatgpt.com