Jurnal Pendidikan Teknik Mesin Vol 26 No. 01 June 2026 (22-31)

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

BIOMECHANICAL ANALYSIS OF ANGLED ABUTMENTS IN POSTERIOR MAXILLA USING FINITE ELEMENT METHOD

Fadhil Putra Ramadhan[1], Kriswanto[1], Kaleb Priyanto[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: fadhilputra24@students.unnes.ac.id

Abstract

This study evaluates the biomechanical effects of angulated abutments (15°, 17°, and 20°) in posterior maxillary implants using three-dimensional Finite Element Analysis (FEA). Due to low bone density and complex anatomy, the posterior maxilla presents a high implant failure risk. Yet, the impact of specific abutment inclinations on internal implant components remains underexplored. Models of D4 bone with bonded implants were created using Autodesk Inventor and simulated in ANSYS under a 276 N vertical load. Von Mises stress, strain, and safety factors were analyzed for abutment and screw components. Results showed increasing angulation significantly elevated stress and strain, especially in screws, though safety factors remained above the 1.5 clinical threshold. Stress concentration occurred at the abutment's U-profile and screw head. These findings suggest angled abutments up to 20° remain biomechanically safe but highlight the need for careful planning to prevent long-term failure.

Key words: Angulated abutment, finite element analysis, dental implant, posterior maxilla, stress distribution

INTRODUCTION

Dental implants have emerged as a globally utilized therapeutic option to replace lost teeth. The use of dental implants has significantly increased, totaling nine million individuals over the last decade (Sartoretto et al., 2023). Dental implants have numerous benefits compared to alternative techniques for repairing absent teeth, including enduring stability and enhanced quality of life. A meta-analysis indicated that the average durability rate of implants was 92% in prospective trials and 88% in retrospective studies over 20 years (Kupka et al., 2024). Prospective research demonstrated that 97.1% of implants retained functionality after 63 months, significantly improving patients' quality of life. This effect increased chewing efficiency and enhanced speech capability post-treatment (Hocková et al., 2025). Although dental implants are commonly used, the likelihood of implant failure from biomechanical issues is significantly heightened in some areas of the mouth cavity, especially in the posterior maxilla.

Wu et al. (2021) reported that the failure rate of dental implants in the posterior maxilla was 2.73 times greater than that of implants in other jaw regions. This condition is related to diminished bone volume in the posterior maxilla resulting from post-extraction bone resorption. The maxillary sinus above this region adds to its anatomical complexity (M. King & Schofield, 2023). Moreover, the posterior maxilla demonstrates the lowest bone density relative to other areas of the human jaw. Misch (2008) classifies the posterior maxilla as D4-density bone, defined by robust cancellous bone and a slender cortical layer. Moreover, the burden experienced by this region is very significant (Edmonds & Glowacka, 2020). One option to mitigate this issue is the utilization of an angled abutment (Cavallaro & Greenstein, 2011). However, the use of angled abutments presents certain challenges.

Lin et al. (2025) asserted that using an angled abutment alters the occlusal stress from an axial to an angular direction. This transition may result in an uneven distribution of pressure and concentration within the implant's structural integrity. Kasari et al. (2023) clarified that this focused pressure may lead to microfractures and bone resorption surrounding the implant. The findings indicate that the design of angled abutments requires a thorough assessment to reduce the likelihood of structural failure of implants from the design phase. Performing a biomechanical assessment of angled abutment designs requires applying dependable and quantifiable techniques.

The finite element analysis (FEA) method is the dominant technique for assessing biomechanics. This method is deemed conservative as it permits actions that do not require direct physical interaction with the human body (Hong & Choi, 2024). This approach may conduct thorough and precise assessments of the intricate human anatomical structure (Shivakumar et al., 2021). It is essential to determine the load capability of dental implant components without failure (Koletsi et al., 2019). Therefore, a safety factor analysis is essential to assess the structural integrity of dental implants, encompassing abutments and screws. The safety factor is the yield strength ratio to maximum stress (Budynas & Keith Nisbett, 2015).

Numerous research has examined the impact of differences in abutment angle on biomechanical behavior. Uppalapati et al. (2023) found that increasing the abutment angle from 0 to 25 degrees increased von Mises stress levels in cortical bone. A study by Bhapkar et al. (2022) indicated that an increase in the abutment angle correlates with heightened stress at the implant-abutment interface. Moreover, Chen et al. (2024) discovered that the inclination of the abutment and the implant insertion angle affect stress, with greater angles leading to elevated stress levels.

Moreover, additional research has utilized safety factor analysis to assess the structural integrity of dental implants. Cantó-navés et al. (2021) performed a safety factor analysis of dental implants subjected to dynamic loading conditions. Furthermore, a study by Menacho-Mendoza et al. (2022) assessed the safety factor in three models of platform switching and internal connections subjected to axial and oblique loading. Kriswanto et al (2025) thoroughly assessed the safety factor in implants utilizing different material combinations and thermal stresses.

Much research has looked at how dental implants work biomechanically, but not many have examined how varied abutment orientations affect biomechanics and safety issues. Also, past research has not looked at the consequences of certain inclination angles—15°, 17°, and 20°—in a consistent way within the same analytical framework. This study fills in the gaps by looking at how these changes in the angle of the abutment (shown in Figure 1) affect the biomechanical performance of the abutment and screw. It also uses a three-dimensional finite element analysis (FEA) to check the safety factor under realistic loading circumstances. This study differs from others because it focuses on changes in abutment inclination as a primary research parameter, which has not been looked at in depth before. This study aims to help doctors choose the best abutment designs for posterior maxillary implants in a helpful way.

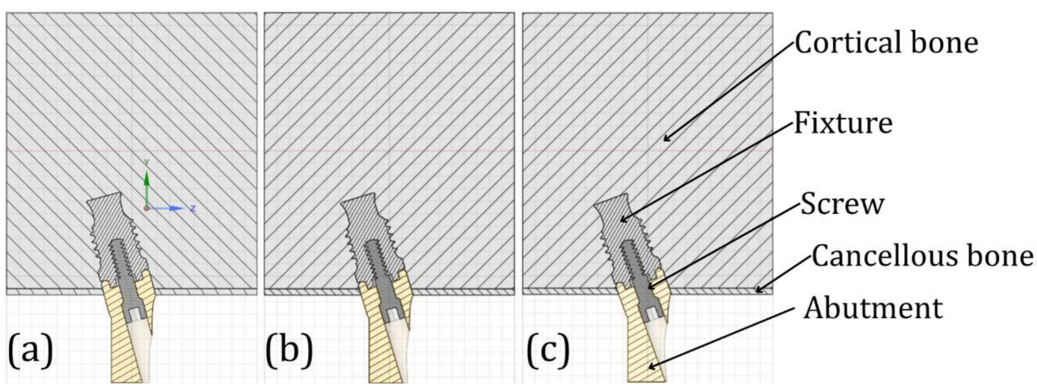


Figure 1. Implant model and structures: (a) 15°; (b) 17° ;(c) 20°

METHODS

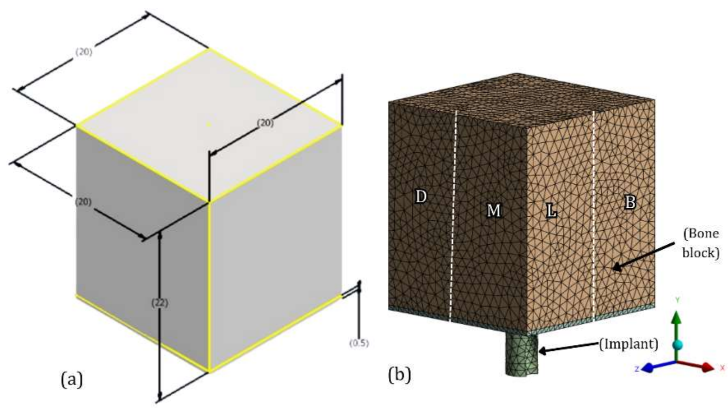
This study employed three-dimensional finite element analysis (FEA) to evaluate the biomechanical behavior of dental implant components with varying abutment inclinations in the posterior maxilla. The first molar's posterior maxilla bone block model was created using Autodesk Inventor Professional 2025 software. The implant structures used were abutment, screw, and fixture; the bone block was modeled into cancellous and cortical. The dental implant fixture design refers to the commercial design of Brånemark System MkIII TiUnite from Nobel Biocare (Nobel Biocare, 2024). The abutment design was based on the Nobel Biocare Replace® Titanium Angled Abutment design. The abutment was designed with 15°, 17°, and 20° inclinations, selected based on clinical use and manufacturer references. The 15° angulation was based on the Nobel Biocare Replace® Titanium Angled Abutment, while the 17° and 20° angulations were adopted from other commercially available systems (BioHorizons, 2020; Osstem Implant Co., 2023).

The three-dimensional model was assumed to be the posterior bone of the maxilla, precisely at the first molar region. The bone model was designed as a block with dimensions of 20 mm (buccolingual) × 20 mm (mesiodistal) × 22 mm (apical-coronal) (Figure 2), based on dimensions from a previous research model (Alqahtani et al., 2023). The bone structure was divided into two layers: the cortical (0.5 mm thickness) and the cancellous at the top. The bone in this study was classified as Type D4, based on Misch's classification of bone density, which is characterized by a very thin cortex and smooth cancellous in the interior (Misch, 2008). The coordinate system used in the modeling refers to the anatomical orientation of the tooth as follows: X-axis points from mesial to distal, Z-axis points from buccal to lingual, and Y-axis. Points upward (coronal-apical direction) (Figure 2). In this study, the implants were placed with an inclination toward the distal direction.

The implant structure consists of three main components: fixture, abutment, and screw (Figure 3). The fixture has a V-shape thread, 4 mm diameter, 7 mm length, 0.6 mm thread pitch, and 0.3 mm thread depth. The abutment includes variations in tilt angles of 15°, 17°, and 20°. In this study, three models were used (Table 1), namely Model 1, Model 2, and Model 3, each of which has a V-shape thread, 4 mm diameter, 7 mm length, 0.6 mm thread pitch, and 0.3 mm thread depth, with the only difference being the abutment tilt angles of 15°, 17°, and 20°, respectively.

Table 1. Implant Model Classification

Name	Thread design	Thread pitch (mm)	Thread depth (mm)	FixtureDiameter (mm)	FixtureLength (mm)	Abutment angulation (°)
Model 1	V-Shape	0.6	0.3	4	7	15
Model 2	V-Shape	0.6	0.3	4	7	17
Model 3	V-Shape	0.6	0.3	4	7	20



M: Mesial; D: Distal B: Buccal; L: Lingual

Figure 2. Entire geometry: (a) bone block dimensions; (b) anatomical directions

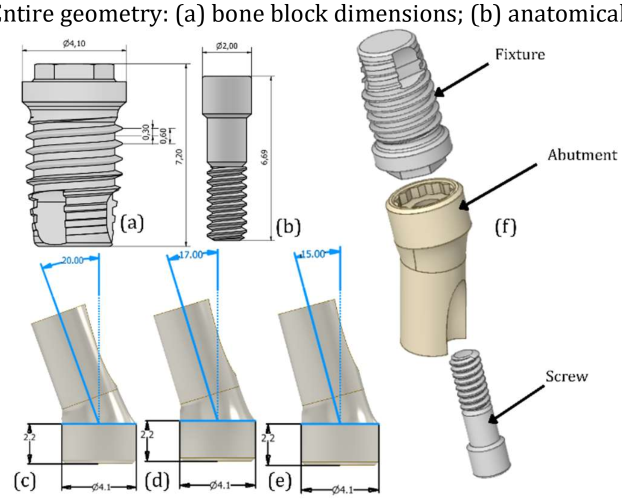


Figure 3. Dental implant configurations: (a) fixture; (b) screw; (c) abutment 15°; (d) abutment 17°; (e) abutment 20°; (f) implant assembly

The general equations in finite element analysis (FEA) are derived from the principle of force equilibrium and can be expressed in matrix form as shown in Equation 1. The matrix {K} is defined as the global stiffness matrix representing the overall stiffness of the system, {d} is the nodal displacement vector indicating the displacement at each node due to the applied load, and {F} is the nodal force vector containing the external forces acting on each node (Logan, 2012).

$$\{F\}=K\{d\} \quad (1)$$

In evaluating stress and strain distributions in the model, this study employs the Von Mises stress approach. The Von Mises equivalent stress is calculated to evaluate the possibility of yielding in ductile materials due to multiaxial loading by comparing the equivalent stress value with the material's yield stress using the distortion energy theory approach, as shown in Equation 2 (Budynas & Keith Nisbett, 2015). Additionally, the Von Mises equivalent strain shown in Equation 3 is also analyzed to evaluate the distribution of equivalent strain as a representation of the total deformation that occurs. This equivalent strain describes the total distortional deformation under three-dimensional stress conditions, incorporating normal and shear strain components, and considering the Poisson's ratio (v) as a correction for material elasticity. This value is used as an initial indicator of plastic deformation (ANSYS, n.d.-a).

This study also evaluates the safety factor in mechanical design, defined as the ratio between material strength and the parameter causing failure, and generally expressed as the design factor (nd) (Equation 4). The basic equation has shown in Equation 4. Where S is the material strength against a specific failure mechanism, and σ or τ is the normal or shear stress acting on the component. The value of nd indicates the safety margin against failure; the higher the value, the safer the design (Budynas & Keith Nisbett, 2015). In this study, the minimum safety factor used is 1.5 (ANSYS, n.d.-b; Cantó-navés et al., 2021).

$$\sigma_{vm} = \sqrt{\frac{1}{2}[(\sigma_1 - \sigma_2)^2 + (\sigma_2 - \sigma_3)^2 + (\sigma_3 - \sigma_1)^2]} \quad (2)$$

$$\varepsilon_{eqv} = \left(\frac{1}{1+\nu}\right)\sqrt{\frac{\left(\varepsilon_{xx} - \varepsilon_{yy}\right)^2 + \left(\varepsilon_{yy} - \varepsilon_{zz}\right)^2 + (\varepsilon_{zz} - \varepsilon_{xx})^2 + 6\left(\varepsilon_{xy}^2 + \varepsilon_{yz}^2 + \varepsilon_{zx}^2\right)}{2}} \quad (3)$$

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

Simulations were performed using ANSYS Workbench 2023 R1 software. The model was exported to ANSYS for meshing and simulation. The elements used were 10-node tetrahedral elements (SOLID187). The meshing process used an element size of 0.74 mm. Mesh convergence testing was performed to ensure the reliability of the simulation results, with the convergence criterion being a von Mises stress difference between consecutive models of less than 5% (Table 2). Element quality in the meshing process was evaluated using the skewness parameter. The average skewness value of the three simulation models used in this study is 0.42546 ± 0.23092. This value falls within the range of 0.25–0.5, which is classified as good (ANSYS, n.d.-b). The meshing process produced the following number of elements and nodes: Model 1 has 115,439 nodes and 72,341 elements, Model 2 consists of 116,977 nodes and 73,310 elements, while Model 3 has 120,080 nodes and 75,395 elements.

The analysis was performed with the Static Structural module. As shown in Figure 4, fixed supports were applied to all sides of the bone block, except the bottom surface, which was left unconstrained. A vertical load of 276 N was applied axially to the occlusal surface of the abutment, corresponding to the maximum human bite force in the first molar area (Srinivasan et al., 2024). All interfaces between the implant components and between the implant and bone were assumed to be bonded contacts, representing a state of complete osseointegration with no relative displacement between surfaces. The parameters analyzed include the Von Mises stress distribution and the equivalent strain and total deformation in both components of the implant system (abutment and screw) to evaluate the effect of varying the angle of inclination of the abutment. The implant structure in this study uses Ti-6Al-4V alloy material, which is modeled as a homogeneous, isotropic, and linearly elastic. The bone material is modeled as orthotropic. The material parameters are presented in Table 3.

Table 2. Convergence test

Mesh size (mm)	Number of Elements	Number of Nodes	σ_{Von} Abutment (Mpa)	σ_{Von} Screw (Mpa)	Difference (%) Abutment	Difference (%) Screw
0.75	67623	107895	104.62	111.39	4.02%	0.46%
0.74	72341	115439	104.48	107.53	-0.20%	-0.13%
0.73	69530	111299	107.18	111.9	0.64%	2.58%

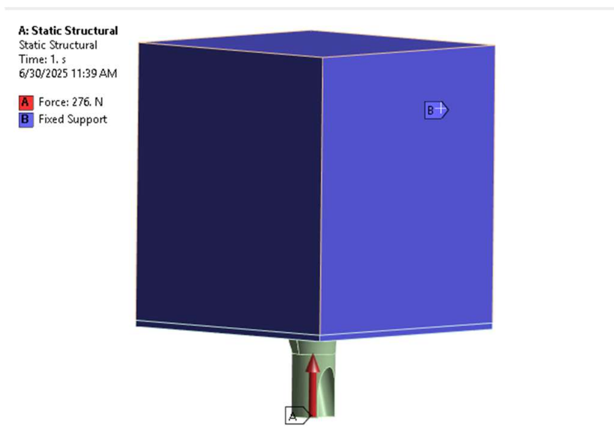


Figure 4. Boundary Conditions

Table 3. Material properties

Component of Model	Young's Modulus (MPa)	Shear Modulus (MPa)	Poisson's Ratio	Yield Strength (Mpa)	References
Cortical	E_x = 12,600	G_{xy} = 4850	ν_{xy} = 0.300	-	(Alqahtani et al., 2023)
	E_y = 12,600	G_{yz} = 5700	ν_{yz} = 0.253	-	
	E_z = 19,400	G_{xz} = 5700	ν_{xz} = 0.253	-	
Cancellous	E_x = 1148	G_{xy} = 68	ν_{xy} = 0.055	-	
	E_y = 210	G_{yz} = 68	ν_{yz} = 0.010	-	
	E_z = 1148	G_{xz} = 434	ν_{xz} = 0.322	-	
Implant structures	113.8	-	0.342	880	(ASM Aerospace Specification Metals Inc, n.d.)

RESULTS AND DISCUSSION

This section outlines the findings from the finite element analysis, simulation results demonstrated that an increase in abutment angulation from 15° to 20° corresponded with a substantial rise in von Mises stress in both the abutment and screw components (Table 3). On the abutment, maximum stress increased from 104.48 MPa at 15°, to 127.74 MPa at 17°, and reached 133.82 MPa at 20°, representing an overall increase of approximately 28.1%. On the screw, the increase was more pronounced, with stress rising from 107.53 MPa at 15°, to 139.89 MPa at 17°, and sharply peaking at 253.66 MPa at 20°, a total increase of 136% (Figure 5). The graphical trend in Figure 5 clearly illustrates a steeper incline in stress for the screw than the abutment. Stress concentration was consistently observed at the U-profile of the abutment and at the outer margin of the screw head across all angulations (Figures 7).

Equivalent strain values also increased consistently with higher abutment angulations (Figure 6). For the abutment, strain rose from 0.0009200 at 15°, to 0.0011228 at 17°, and slightly further to 0.0011777 at 20°, marking an increase of approximately 28%. On the screw, strain increased from 0.0011495 to 0.0015148, and then rose sharply to 0.0022890 at 20°, nearly doubling over the angular range with a total increase of 99.1%. The strain curve in Figure 6 shows a steeper gradient for the screw, indicating greater sensitivity to angulation. Similar to stress distribution, strain concentration was located at the U-profile of the abutment and the outer edge of the screw head (Figure 8).

The factor of safety (FOS) decreased with increasing abutment angulation for both components (Table 4). On the abutment, FOS declined from 8.42 at 15°, to 6.88 at 17°, and to 6.57 at 20°. In the screw, the FOS dropped from 8.18 to 6.29 and reached its lowest value of 3.46 at 20°.

Table 3. Maximum von Mises Stress and Strain on Abutment and Screw at Varying Abutment Angulations

Abutment Angulation (°)	Max σ_{Von} (Mpa)		Max ε_{Von}	
	Abutment	Screw	Abutment	Screw
15	104.48	107.53	0.0009200	0.0011495
17	127.74	139.89	0.0011228	0.0015148
20	133.82	253.66	0.0011777	0.0022890

Tabel 4. Factor of Safety

Abutment Angulation (°)	Factor of Safety Abutment (Max 1.5)	Factor of Safety Screw (Max 1.5)
15	8.42	8.18
17	6.88	6.29
20	6.57	3.46

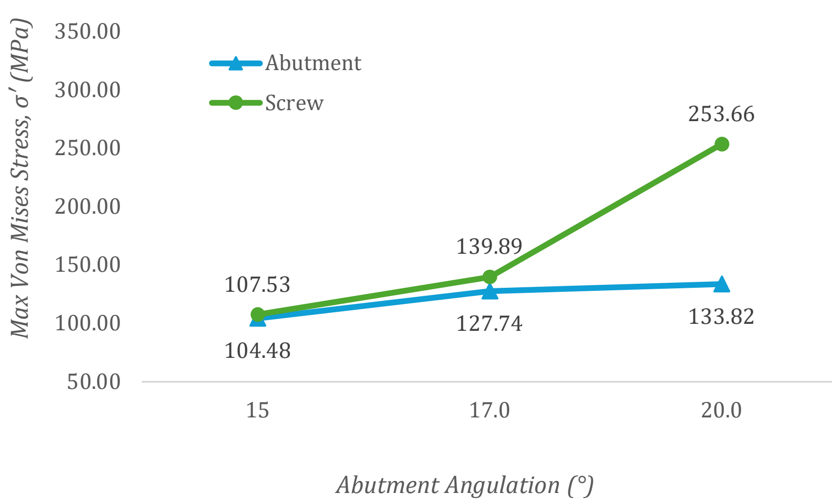


Figure 5. Graph of Maximum von Mises Stress on Abutment and Screw

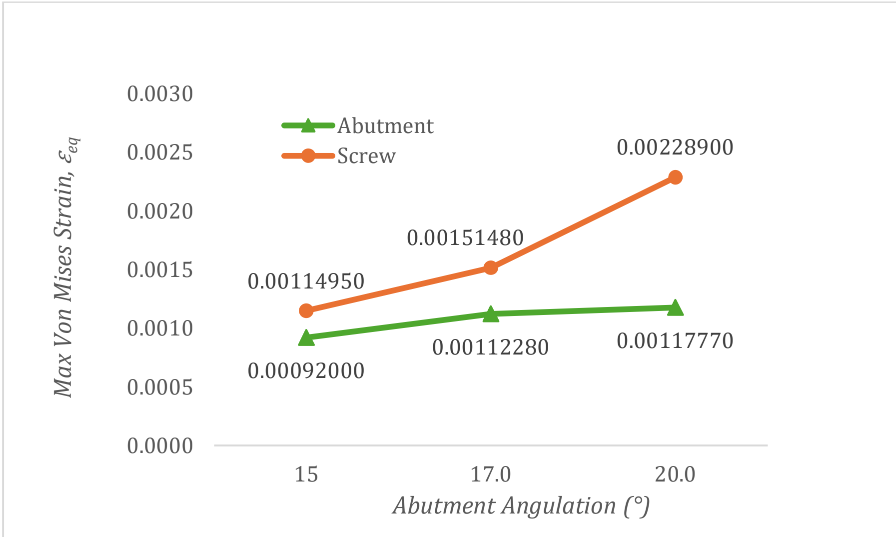


Figure 6. Graph of Maximum von Mises Strain on Abutment and Screw

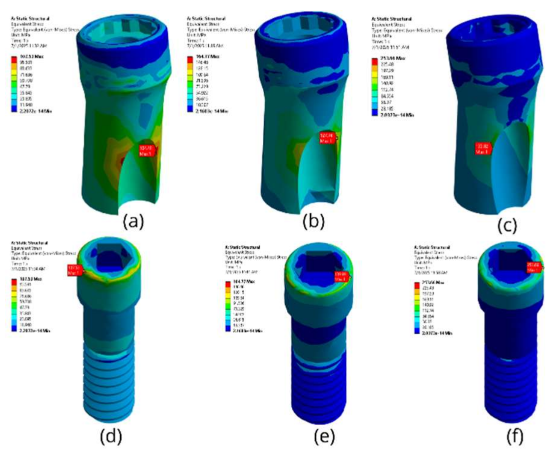


Figure 7. von Mises Stress on the Abutment: (a) 15°; (b) 17°; and (c) 20°; and on the Screw: (d) 15°; (e) 17°; (f) 20°

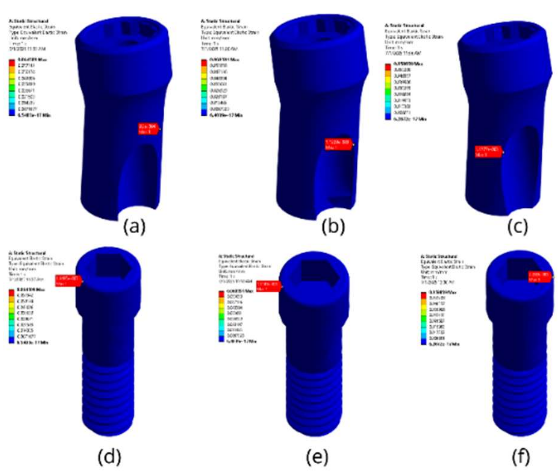


Figure 8. von Mises Strain on the Abutment: (a) 15°; (b) 17°; and (c) 20°; and on the Screw: (d) 15°; (e) 17°; (f) 20°

The simulation findings suggest that all of the examined abutment angle configurations are safe from a biomechanical point of view. The highest stress value in the whole model was found in component screw model 3 (253.66 MPa). This is still significantly below the yield strength of Ti-6Al-4V material, which is about 880 MPa (ASM Aerospace Specification Metals Inc, n.d.). With a safety factor (FoS) of 3.46 > 1.5, as shown in the study by Cantó-navés et al. (2021). This means that even while the stress and strain on the implant system increases when the abutment angle goes up, it stays in the elastic zone and does not seem to be able to deform or break down directly.

Previous research has demonstrated that a higher abutment angle is directly linked to more stress in the implant system. De Faria Almeida et al. (2022) found that angled abutments at 17° and 30° put more stress on implant parts and supporting tissues than straight abutments, especially when the loads were not straight. Korkmaz & Kul (2022) also showed that changing the abutment angle from 17° to 25° put more stress on the prosthetic structure, cortical bone, and trabecular bone, especially when the load was not straight. Uppalapati et al. (2023) and Oswal et al. (2024) also support these results by showing that bigger abutment angles always worsen stress concentration in the implant system.

This study has several limitations that should be noted. The bone model used is a simplified version that does not correctly show how complicated bones are anatomically and how they work mechanically. Models based on CT scans might be more useful for clinical situations (Lisiak-Myszke et al., 2020). Also, employing bonded contact assumptions in the simulation does not match what happens in real life (Kapoor et al., 2021). One problem with this study is that it only used static loading, which does not show how well dental implants will work in the long term (Erdoğdu et al., 2024). As a result, future research should use CT-based bone models, stick to the assumptions about friction contact, and include dynamic loading

CONCLUSIONS

Based on the results of the analysis and discussion, the following conclusions can be drawn:

1. Increasing the abutment angle from 15°, 17°, to 20° resulted in higher von Mises stress by 28.1% on the abutment and 135.9% on the screw, with stress concentration located at the U-profile area of the abutment and the outer edge of the screw head.
2. Maximum strain followed a similar trend, increasing by 28.3% on the abutment and 99.1% on the screw, indicating a higher risk of elastic deformation under oblique loading.
3. All abutment angle configurations maintained a safety factor above the clinical minimum threshold of 1.5, with the lowest value being 3.46 at the 20° screw, confirming their safe biomechanical application in posterior maxilla cases with proper biomechanical planning.

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). https://doi.org/10.1186/s12903-023-03370-8

ANSYS, Inc. (n.d.-a). *Chapter 17: Postprocessing in ANSYS Mechanical APDL 2024 R2 Theory References.* Retrieved July 3, 2025, from https://ansyshelp.ansys.com/public/account/secured?returnurl=////////Views/Secured/corp/v242/en/ans_thry/thy_post12.html

ANSYS, Inc. (n.d.-b). *Skewness*. Retrieved July 3, 2025, from 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*.

Bhapkar, Dr. K., MV, Dr. S. K., Kumar, Dr. R., Saxena, Dr. R., Khandelwal, Dr. T., & Rana, Dr. V. (2022). A comparitive evaluation of stress distribution around implant with different abutment material at different abutment angulation: A finite element analysis study. *International Journal of Applied Dental Sciences*, *8*(2), 309–317. https://doi.org/10.22271/oral.2022.v8.i2e.1524

BioHorizons. (2020). *Prosthetic Solutions Product Catalog*.

Budynas, R. G., & Keith Nisbett, J. (2015). *Shigley's Mechanical Engineering Design 10th Edition*.

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

Cavallaro, J., & Greenstein, G. (2011). Angled implant abutments: A practical application of available knowledge. *Journal of the American Dental Association*, *142*(2), 150–158. https://doi.org/10.14219/jada.archive.2011.0057

Chen, P., Zhang, J., Yao, J., Hu, F., Song, L., & Yu, Y. (2024). Effect of angled abutments in the posterior maxillary region on tilted implants: a 3D finite element analysis. *Medical and Biological Engineering and Computing*, *62*(8), 2585–2597. https://doi.org/10.1007/s11517-024-03081-4

de Faria Almeida, D. A., Verri, F. R., Lemos, C. A. A., de Souza Batista, V. E., Santiago Júnior, J. F., Rosa, C. D. D. R. D., Noritomi, P. Y., & Pellizzer, E. P. (2022). Effect of Splinting of Tilted External Hexagon Implants on 3-Unit Implant-Supported Prostheses in the Posterior Maxilla: A 3D Finite Element Analysis. *Journal of Prosthodontics*, *31*(8), 697–704. https://doi.org/10.1111/jopr.13460

Edmonds, H. M., & Glowacka, H. (2020). The ontogeny of maximum bite force in humans. *Journal of Anatomy*, *237*(3), 529–542. https://doi.org/10.1111/joa.13218

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*, *14*(10). https://doi.org/10.3390/jpm14101040

Hocková, B., Slávik, R., Poruban, D., Bonfante, E. A., Ewers, R., Stebel, A., & Cheng, Y. C. (2025). Quality of life of patients receiving short dental implants in microvascular free flaps: a five year prospective study. *Scientific Reports*, *15*(1). https://doi.org/10.1038/s41598-025-01012-3

Hong, M. H., & Choi, H. (2024). Three-Dimensional Finite Element Analysis of Stress Distribution in Dental Implant Prosthesis and Surrounding Bone Using PEEK Abutments. *Biomimetics*, *9*(8). https://doi.org/10.3390/biomimetics9080472

Kapoor, S., Rodrigues, S., Mahesh, M., Shetty, T., Pai, U., Saldanha, S., Hedge, P., & Shenoy, S. (2021). Evaluation of Stress Generated with Different Abutment Materials and Angulations under Axial and Oblique Loading in the Anterior Maxilla: Three-Dimensional Finite Element Analysis. *International Journal of Dentistry*, *2021*. https://doi.org/10.1155/2021/9205930

Kasari, M. S., Yousefi, H., Shirazi, A. E., & Geramy, A. (2023). Effect of bone quality and osseointegration on stress distribution in the bone tissue surrounding dental implant: a finite element analysis. *Journal of Osseointegration*, *15*(2), 106–112. https://doi.org/10.23805/JO.2023.552

Koletsi, D., Iliadi, A., Eliades, T., & Eliades, G. (2019). In vitro simulation and in vivo assessment of tooth wear: A meta-analysis of in vitro and clinical research. In *Materials* (Vol. 12, Issue 21). MDPI AG. https://doi.org/10.3390/ma12213575

Korkmaz, I. H., & Kul, E. (2022). Investigation of the Type of Angled Abutment for Anterior Maxillary Implants: A Finite Element Analysis. *Journal of Prosthodontics*, *31*(8), 689–696. https://doi.org/10.1111/jopr.13462

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

Kupka, J. R., König, J., Al-Nawas, B., Sagheb, K., & Schiegnitz, E. (2024). How far can we go? A 20-year meta-analysis of dental implant survival rates. *Clinical Oral Investigations*, *28*(10), 541. https://doi.org/10.1007/s00784-024-05929-3

Lin, D., Wen, Z. H., Guo, X. N., & Zhang, Y. (2025). Stress distribution on the components of multi-unit abutment with different angulation under lateral force: a finite element analysis. *Clinical Oral Investigations*, *29*(5). https://doi.org/10.1007/s00784-025-06306-4

Lisiak-Myszke, M., Marciniak, D., Bieliński, M., Sobczak, H., Garbacewicz, Ł., & Drogoszewska, B. (2020). Application of finite element analysis in oral and maxillofacial surgery-A literature review. In *Materials* (Vol. 13, Issue 14). MDPI AG. https://doi.org/10.3390/ma13143063

M. King, E., & Schofield, J. (2023). Restoratively driven planning for implants in the posterior maxilla - Part 1: alveolar bone healing, bone assessment and clinical classifications. *British Dental Journal*, *235*(8), 585–592. https://doi.org/10.1038/s41415-023-6391-7

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

Misch, C. E. . (2008). *Contemporary implant dentistry*. Mosby Elsevier.

Nobel Biocare. (2024). *Nobel Biocare Product Catalog 2024/2025*.

Osstem Implant Co., Ltd. (2023). *Osstem – 2023 Product Catalog*.

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

Sartoretto, S. C., Shibli, J. A., Javid, K., Cotrim, K., Canabarro, A., Louro, R. S., Lowenstein, A., Mourão, C. F., & Moraschini, V. (2023). Comparing the Long-Term Success Rates of Tooth Preservation and Dental Implants: A Critical Review. In *Journal of Functional Biomaterials* (Vol. 14, Issue 3). MDPI. https://doi.org/10.3390/jfb14030142

Shivakumar, S., Kudagi, V. S., & Talwade, P. (2021). Applications of finite element analysis in dentistry: A review. In *Journal of International Oral Health* (Vol. 13, Issue 5, pp. 415–422). Wolters Kluwer Medknow Publications. https://doi.org/10.4103/JIOH.JIOH_11_21

Srinivasan, M., Tahoury, B., Takeshita, L., Moreira, Y. M., Pavan, J. S. R., & Leles, C. R. (2024). Correlates and Comparability of Two Devices for Measuring Bite Force in Dentate Health Volunteers. *Journal of Oral Rehabilitation*. https://doi.org/10.1111/joor.13914

Uppalapati, V., Kumar, S., Aggarwal, R., Bhat, I., Munaganti, J., & Khan, S. (2023). Three-dimensional Finite Element Stress Pattern Analysis in Bone around Implant-supported Abutment with Different Angulations under Axial and Oblique Load. *Journal of Contemporary Dental Practice*, *24*(1), 16–20. https://doi.org/10.5005/jp-journals-10024-3464

Wu, X., Chen, S., Ji, W., & Shi, B. (2021). The risk factors of early implant failure: A retrospective study of 6113 implants. *Clinical Implant Dentistry and Related Research*, *23*(3), 280–288. https://doi.org/10.1111/cid.12992