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

COMPARATIVE ANALYSIS OF DENTAL IMPLANT DIAMETERS IN VON MISES DISTRIBUTION: IMPLICATIONS FOR CLINICAL DESIGN

Gunagung Syahrul Nizam Sulkan[1], Kriswanto[1], Ruben Bayu Kristiawan[1] ,Jamari[2], Athanasius Priharyoto Bayuseno[2]

[1] Semarang State of University , Semarang
[2] Diponegoro of University, Semarang
Email: syahrul145@students.unnes.ac.id

Abstract

The loss of anterior dental has a significant impact on masticatory function and facial aesthetics, making dental implants a widely used rehabilitative solution. The success of implants is greatly influenced by geometric design, particularly the diameter of the fixture, which plays a role in distributing load. This study aims to analyze the influence of variations in implant diameter on stress distribution, strain, and safety factor in the screw and abutment areas using a three-dimensional Finite Element Analysis (FEA) approach. Three implant models with fixture diameters of 3.75 mm, 4.00 mm, and 5.00 mm were simulated using a vertical physiological load of 130.5 ± 73.6 N, assuming Ti-6Al-4V material and orthotropic properties of the mandibular bone. The results showed that Model 3 (5.00 mm diameter) had the most balanced stress distribution in screw, hight performance in von mises strain and the safety factor value at the abutment (12.22), while Model 2 exhibited the lowest stress in abutment and hight performance in safety factor (12.25) . Model 1 had the lowest biomechanical performance with the highest stress and strain and the lowest abutment safety factor (10.61). It can be concluded that variations in fixture diameter significantly affect the mechanical performance of the implant system, and a diameter of 5.00 mm is recommended as the most optimal design to support long-term strength and stability.

Key words: Von mises, Safety Factor, Finite Element Analisys, Implant Dental

INTRODUCTION

A lack of public awareness of the importance of dental care generally causes dental decay. If it occurs in the front dental, it will affect the appearance of the face (Aldabeeb et al., 2024). Dental decay, primarily due to caries, is still a public health problem in Indonesia. The data results of the 2018 Riskesdas noted that 57.6% of the population experienced dental and oral problems, but only 10.2% received medical treatment. Especially in early childhood, cavities are very high (93%), indicating the urgency of long-term solutions (Riset Kesehatan Dasar (Riskesdas), 2018). There are many ways to overcome cavities, from filling dental cavities and installing dentures to dental implants.

Dental implants are becoming an increasingly popular restorative solution because they can restore masticatory function and aesthetic appearance more optimally than conventional dentures (Pjetursson et al., 2014). Implants consist of three main components: the fixture implanted in the jawbone, the abutment as a connector, and the crown as the prosthetic part visible above the gums. Of the three implant components, the crown plays a vital role as it receives and spreads chewing forces evenly (Tribst et al., 2016). The successful long-term use of an implant is greatly influenced by the process of osseointegration, which is the structural and functional attachment between bone and the implant surface that ensures permanent stability (Reif et al., 2022). In clinical studies, the 10-year survival rate of implants reached 99.4% in healthy individuals and 95.9% in patients with systemic conditions (Sodnom-Ish et al., 2023).

Implant materials must have high biocompatibility to prevent premature failure and inflammation. In addition, the material must have good mechanical properties, resistance to friction (tribology), and be able to stimulate the growth of surrounding tissue (Daniel et al., 2024). Depending on how they interact with biological tissues, implant materials are generally categorized into three main types: biotolerant, bioinert, and bioactive (Stoykov et al., 2022). Biotolerant materials like stainless steel and cobalt-chromium alloys, may provoke tissue irritation, forming fibrous encapsulation (Sotova et al., 2023). In contrast, bioinert materials including titanium, alumina, zirconia, and tantalum can form stable physical attachments to bone without initiating chemical reactions. Meanwhile, bioactive materials such as calcium phosphate, bio-glass, and glass ceramics can form direct chemical bonds with bone tissue, thereby enhancing biological integration (Sotova et al., 2023).

In addition to the material, the fixture, abutment, and screw design also affect the stress distribution. The shape and thickness of these elements determine the bite load distribution. Suboptimal design can trigger excessive stress that causes cracks and long-term structural failure (Moreno et al., 2023). Therefore, Von Mises analysis of the screw is needed to analyze the connection between the abutment and the correct fixture to ensure the durability of the implant against biomechanical stress that occurs in daily activities. FEA offers advantages in accurately visualizing stress and strain distributions, with significantly better cost and time efficiency compared to direct experimental methods(Tan et al., 2022; Verma et al., 2024; Zainal Abidin et al., 2022).

In Finite Element Analysis (FEA), meshing is very important because it affects the quality of the convergence test later on. Mesh quality refers to how well the mesh elements fit the physical geometry of the model. A good quality mesh should have uniform elements and not experience excessive deformation. Recent research emphasizes that the larger the implant size, the better the stress distribution. (Ayres et al., 2025) show that Larger diameter abutments tended to generate better stress distribution (Armentia et al., 2022) also found that increasing the diameter of the implant body and platform significantly improves the fatigue life of prosthetic screws. However, excessive increases in abutment screw diameter increase internal stress concentration in the screw itself. Research by (González-Mederos et al., 2025) also highlights the importance of combining diameter, length, and material parameters to support optimal mechanical behavior in implant systems. However, there are still limitations in studies that specifically analyze the influence of implant diameter variations on stress and strain distribution at the abutment–screw interface while considering bone's orthotropic properties. Few studies explicitly integrate a safety factor analysis approach into implant biomechanical simulations. . Safety factor is the yield strength ratio to actual working stress (von Mises), which is important to ensure that the implant structure remains safe against potential overloads or unexpected events. By incorporating safety factor analysis, implant design can be improved, and the risk of long-term failure can be minimized (Cervino et al., 2018).

This study aims to analyze the effect of implant diameter variation on the stress and strain distribution in the screw and abutment areas under native conditions in the anterior mandibular area using a three-dimensional Finite Element Analysis approach. This study will also evaluate the safety factor values for each diameter variation to determine the most optimal implant design for strength and long-term stability. The results of this study are expected to make an important contribution to the development of safe, efficient, and biomechanically durable implant designs.

METHODS

This research was conducted numerically using the Finite Element Analysis (FEA) method to analyze the distribution of stress, strain, and von Mises safety factor for its application using ansys mechanical 2023 R1 software. The finite element method can analyze the structure, shape, load, mechanical properties, and so on in materials, and can analyze stress distribution objectively, accurately, and precisely (Quiudini et al., 2017).

The analysis was performed by processing the numerical simulation results from the finite element software, where von Mises stress was used to determine whether the maximum stress occurring in the structure was below the material's yield limit. Thus, this approach provides a strong foundation for evaluating the safety and feasibility of structural designs against failure due to working loads (Cervino et al., 2018). Here is the von mises (VM) formula according to (Chandrupatla, 1991) :

$$\sigma_{VM} = \frac{1}{\sqrt{2}}\sqrt{(\sigma_x - \sigma_y)^2 + (\sigma_y - \sigma_z)^2 + (\sigma_z - \sigma_x)^2}$$

Where σ_x , σ_y , σ_z are the normal stress in XYZ. Respectively σ_{vm} is the von mises equivalent stress.

The commonly used Safety Factor ranges up to 1.5 for general engineering applications (Implant etc.), depending on uncertainties in loads, environmental conditions, and material properties (Cantó-navés et al., 2021). Kriswanto et al., (2025) conducted a similar investigation, revealing that changes in fixture, abutment, crown materials, and thermal loading significantly affected stress distribution, displacement, and the resulting factor of safety (FOS). Higher values are generally used for high-risk structures or those requiring high reliability, such as in the aerospace and medical industries, including dental and orthopedic implant designs ((Aldabeeb et al., 2024; Cicciù et al., 2015). Here is the Factor of Safety *(F.S)* formula according to (Hibbeler, 2018):

$$F.S. = \frac{\sigma\ Fail}{\sigma\ alllow} = \frac{Yield\ Strenght}{Von\ Mises\ stress}$$

Dental implants consist of three main components: fixture, abutment, and screw. In this study, design variations focused on the fixture, with the abutment and screw dimensions adjusted accordingly. The fixture design was modeled after the Nobel Brånemark System MkIII TiUnite with reverse buttress thread and the abutment cemented using a screw. To replicate the design, the size reference from the catalog found on the web was used, and the manufacturing process was carried out using Inventor Professional 2025 software, resulting in the design shown in Figure 1 and then exported to ANSYS Workbench 2024 in STEP format for analysis. Ti-6Al-4V titanium material was specified for all implant components. The simulation used linear static analysis, with vertical loading and boundary conditions of fixed support in the fixture.

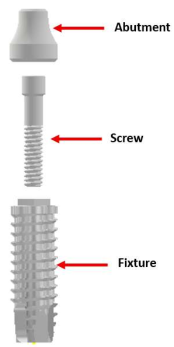


Figure 1. Implant (Fixture , Screw dan Abutment)

Information about the mesh quality of this model is presented in the following table. The skewness value plays a crucial role in evaluating the extent to which the meshing process successfully forms elements according to the expected shape and supports the modeling objectives (You et al., 2006).

Table 1 Skewness mesh quality

Excellent	**Very good**	**Good**	**Accept able**	**Bad**	**Unaccepta ble**
0-0.25	0.25-0.50	0.50-0.80	0.80-0.94	0.95-0.97	0.98-1.00

In this study, experiments were conducted on three models: Model 1 with a fixture length of 11, diameter of 3.75, and taper of 7, with abutments and screws matching the model dimensions. Model 2 with a fixture length of 11, diameter of 4, and taper of 7, with abutments and screws matching the model's dimensions. Model 3 with a fixture length of 11, diameter of 5, and taper of 7, with abutments and screws matching the model's dimensions. The implant materials (fixture, abutment, screw) are assumed to be made of titanium alloy Ti-6Al-4V with isotropic elastic properties , The model was created using a re-modeling process according to the sizes available in the Nobel Biomark catalog using software Inventor Profesional 2025 .

The simulation was performed with axial loading applied to the anterior mandible at 13.3 ± 7.5 kp = 130.5 ± 73.6 N, as recommended in the literature (Kinra et al., 2017). The loading direction was aligned with the physiological bite direction (vertical downward).

Table 2. Material Properties

Material Properties	**Ti-6Al-4V**
Young Modulus	113.8 GPa
Possion Ratio	0.342
Shear Modulus	44 GPa
Yield Strenght	0.88 Gpa

Boundary conditions were applied to the interior of the mandible in the form of fixed supports to maintain model stability during simulation. Meshing was performed automatically using quadratic tetrahedral elements, and convergence checks < 5% were performed to ensure stable simulation results without significantly increasing computation time. Research with error values below 5% can be continued (Aversano et al., 2021). Figure 2 shows the simulation setup where the fix support is on the part labeled A, which is on the fixture thread part, while the bite force is applied on Label B (abutment part).

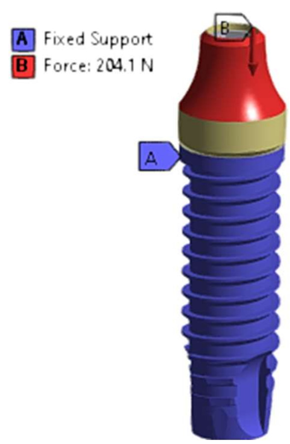


Figure 2. Fix support and Force Implant

Convergence tests were conducted through an independent meshing approach to evaluate the effect of element size on Von Mises stress. The optimal mesh was selected based on the smallest variation (<5%) in the simulated stresses in table 3.

Type Mesh	Mesh Sizing (mm)	Number of Nodes	Number of Elements	Von Mises Stress Screw (MPa)	Von Mises Stress Abutment (MPa)	Persentage Variation Abutment (%)	Persentage Variation Screw (%)
Quadratic Tetrahedral	0.59	254300	157683	40.801	80.869	-	-
Quadratic Tetrahedral	0.58	265802	165501	39.97	82.93	-2.04	2.55
Quadratic Tetrahedral	0.57	268650	167095	40.28	84.96	0.78	2.45
	% Difference between 0.57- and 0.58 -mm mesh				2.04% and -2.55 %		
	% Difference between 0.57- and 0.56 -mm mesh				0.78 % and 2.45 %		

The model testing with an element size of 0.58 mm, which was obtained after convergence testing to ensure that the stress results did not change significantly even though the number of elements increased.

Table 3. Result Meshing Independent Test

Figure 3. Mesh Implant

This study evaluates the effect of dental implant design variations on Von Mises stress occurring in implant screws. Three design parameters were varied: diameter (3.75 mm, 4.00 mm, and 5.00 mm). The results of finite element analysis simulations show that the combination of these three parameters has a significant effect on the distribution of stress occurring in implant screws.

The mesh quality evaluation results show that the skewness value is in the "very good" category, indicating that the mesh elements have a stable and accurate shape for the numerical simulation process. This ensures that the resulting stress distribution is reliable and representative of the simulated model conditions with the results in Table 4.

Table 4. Result Mesh Skew

Display style	Use Geometry Setting
Default	
Physics Preference	Mechanical
Element Order	Quadratic
Element Size	0.58
Sizing	
Check Mesh Quality	Yes, Errors
Error Limits	Aggressive Mechanical
Target Element	Default
Smoothing	High
Mesh Metric	Skewness
Min	3.815e-003
Max	0.99932
Average	0.48454
Standard Deviation	0.22538

RESULTS AND DISCUSSION

Table 5 Result Simulation

Model	Von Mises Stress (Mpa)		Von Mises Strain (mm)		Factor of safety	
	Screw	Abutment	Screw	Abutment	Screw	Abutment
Model 1	39.97	82.93	3.55E-04	7.52E-04	15	10.611
Model 2	39.19	71.83	3.65E-04	6.46E-04	15	12.251
Model 3	28.7	72.02	2.55E-04	6.38E-04	15	12.218

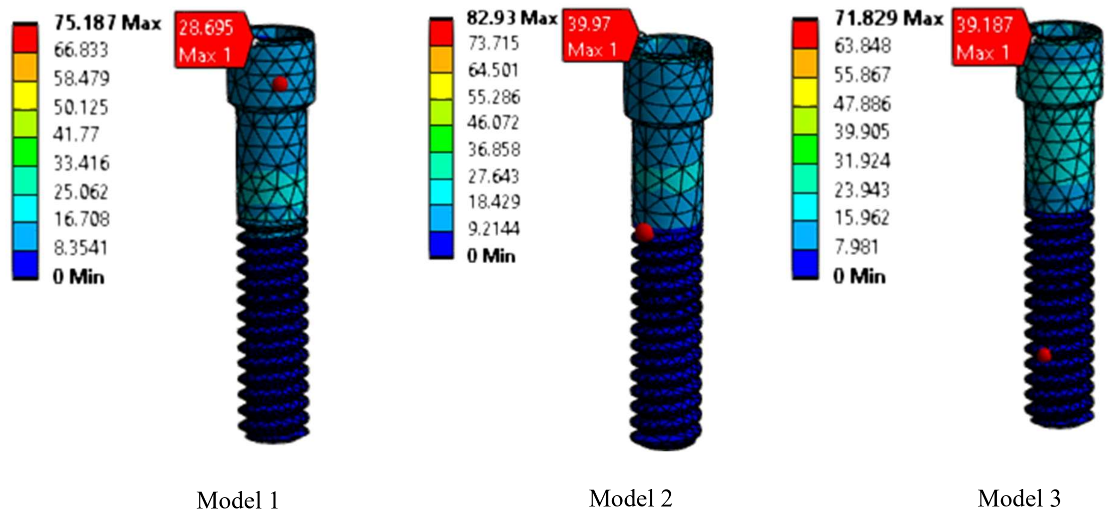


Figure 5. Von Mises Stress Screw at three model

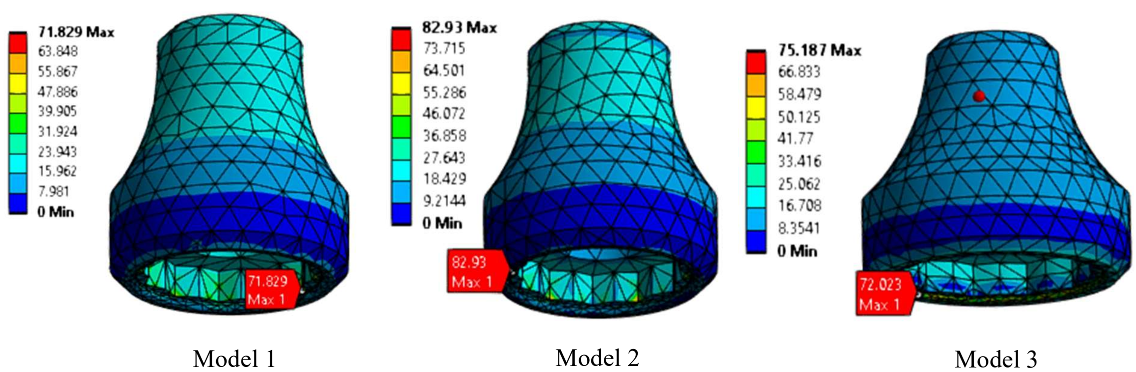


Figure 6. Von Mises Stress Abutment at three Model

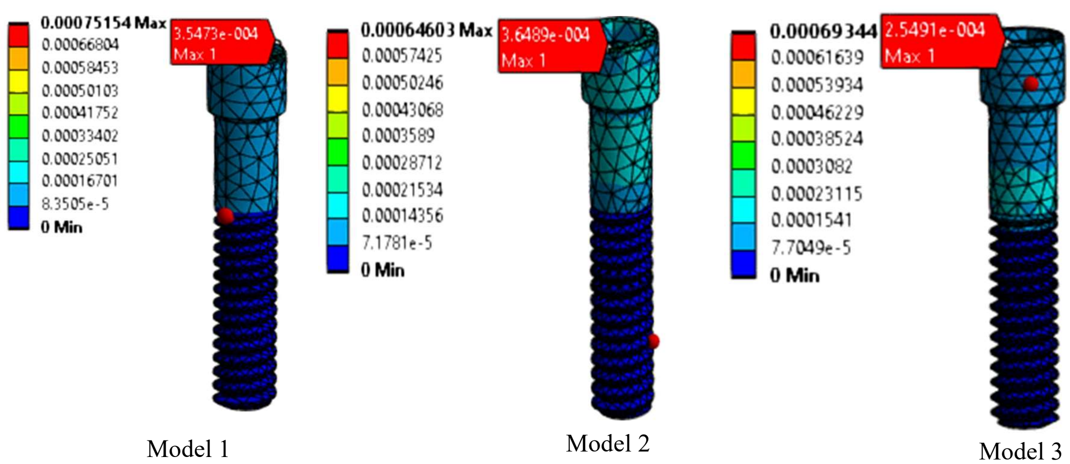
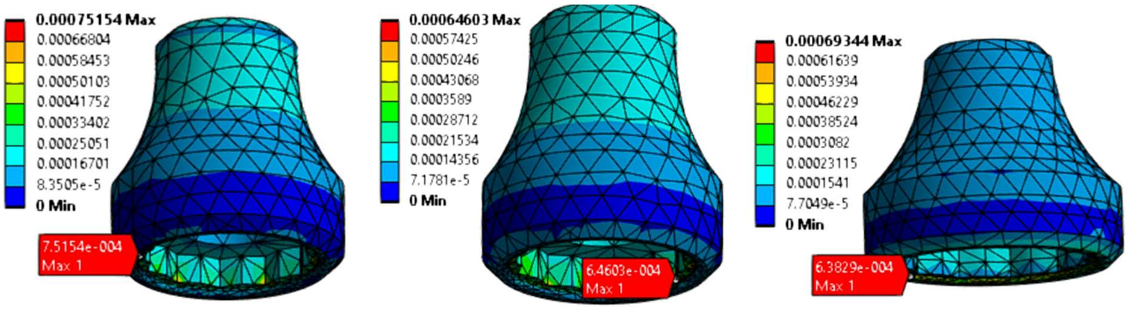
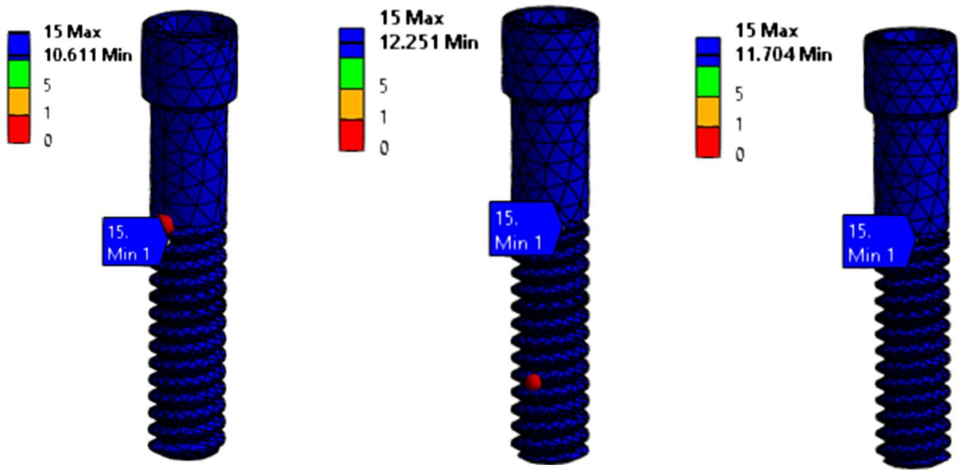


Figure 7. Von Mises Strain Screw at Three Model



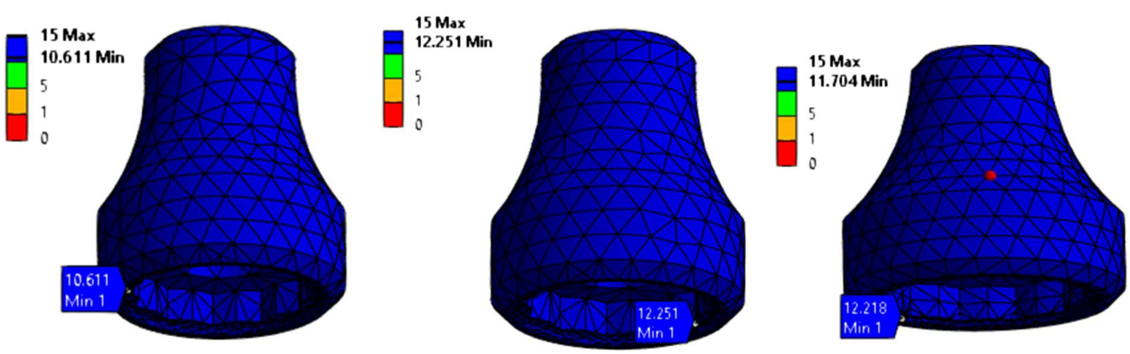
Model 1 Model 2 Model 3

Figure 8. Von Mises Strain Abutment at three model



Model 1 Model 2 Model 3

Figure 9. Factor of Safety Screw at three Model



Model 1 Model 2 Model 3

Figure 10. Factor of Safety Screw at three Model

Simulation results using the finite element method show differences in stress distribution, strain, and safety factor among three dental implant design models. The parameters analyzed include Von Mises Stress (stress), Von Mises Strain (strain), and Safety Factor (safety factor) in the screw and abutment areas. The Von Mises method is widely used in biomechanical studies because it can accurately predict material failure under complex loads (Cervino et al., 2018; Cicciù et al., 2015).

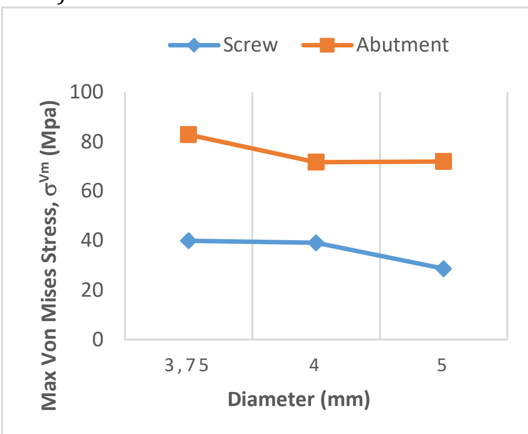


Figure 10. Graph of fixture diameter vs von mises stress

In the screw area, the highest Von Mises stress value was found in Model 1 at 39.9 MPa, while the lowest was in Model 3 at 28.70 MPa. Conversely, in the abutment area, Model 1 showed the highest stress value at 82.93 MPa, while Model 2 had the lowest stress at 71.83 MPa. These stress values indicate that the design of Model 2 is able to reduce the load on the abutment compared to Model 1, but not significantly on the screw section. This aligns with the findings of (Aldabeeb et al., 2024), who stated that variations in implant design can affect stress distribution, the part with the highest stress strain is at the neck (connection between the screw and the fixture).

Von Mises Strain, the highest strain in the screw area was also found in Model 2 at 3.65E-04 mm, while Model 3 had the lowest strain at 2.55E-04 mm. In the abutment area, Model 1 showed the highest strain (7.52E-04 mm), consistent with the high stress values observed, while Model 3 exhibited a lower strain value (6.38E-04 mm), approaching that of Model 2. This indicates a direct relationship between stress and strain in implant design, as discussed by (Yang et al., 2023)in their FEA study on short implants.

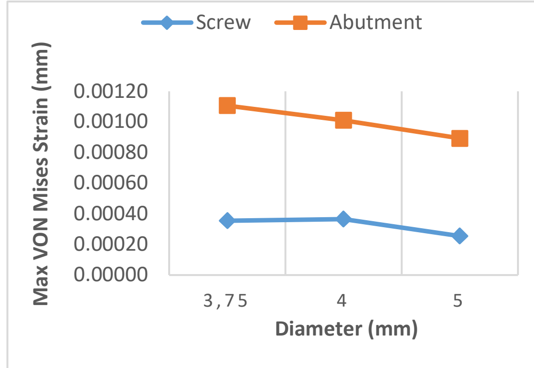


Figure 11. Graph of fixture diameter vs von mises strain

The safety factor in the screw area shows a constant value of 15, indicating that the screw design in all three models remains well below the yield strength limit of titanium material (Nicholson, 2020) However, there is variation in the abutment area. Model 2 has the highest safety factor of 12.25, followed by Model 3 at 12.22, and the lowest is Model 1 at 10.61. A higher safety factor indicates that the design is safer against the risk of structural failure.

CONCLUSIONS

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

1. This study evaluated the effect of implant diameter variation on stress distribution, strain, and safety factors in the screw and abutment areas using a three-dimensional Finite Element Analysis approach. Based on the results of numerical simulations conducted on three models with fixture diameters of 3.75 mm, 4.00 mm, and 5.00 mm, the following conclusions can be drawn:

2. Variations in implant diameter significantly affect stress and strain distribution, particularly in the abutment area. The model with a diameter of 5.00 mm (Model 3) showed the most balanced stress and strain distribution. When combined, von mises on the abutment and screw have lower values than the others, with von mises of 28.7 on the screw and 72.02 on the abutment and strain of 6.46E-04 mm.

3. Model 2 has the highest safety factor value at the abutment, which is 12.251 indicating better structural reliability than Model 1 and Model 3. Meanwhile, the safety factor value at the screw area remains constant at 15 across all models, indicating that the screw design is well below the material failure limit.

4. Model 2 shows the best performance in the abutment area with the lowest stress (71.83 MPa), but its safety factor is slightly lower than Model 2 at the abutment. This indicates that increasing the diameter does not always result in a linear increase in all biomechanical parameters.

5. Model 1 exhibits the lowest biomechanical performance, particularly in the abutment, with the highest stress and strain values and the lowest safety factor (10.61). Although still within safe limits, this design is less optimal than the other two models.

6. The selection of the optimal implant diameter, particularly within the 5.00 mm range, is recommended to achieve a balance between mechanical strength and structural safety, as well as to minimize the risk of long-term failure in the abutment–screw connection system.

7. Considering the stress distribution, strain, and safety factor analysis results, Model 3(diameter 5) can be recommended as the most optimal implant design in this study. The findings of this research are expected to serve as a reference for designing safer, more efficient, and biomechanically durable dental implants. which is in accordance with the statement (Ayres et al., 2025) that the wider the implant size, the better.

8. From the results of all simulations on the screw, the part with the highest stress strain is at the neck (connection between the screw and the fixture), as well as at the abutment, the highest point of strain and stress is at the bottom (connection between the abutment and the fixture).

REFERENCE

Aldabeeb, D., Alofi, A., Alfaran, R., Salam, A., Alsuhaibani, L., & Alanazi, M. (2024). The Psychological Impact of Anterior Teeth Caries and Treatment Needs Among Adults at King Saud University Clinics in Riyadh, Saudi Arabia. *Cureus*, *16*(8). https://doi.org/10.7759/cureus.68115

Armentia, M., Abasolo, M., Coria, I., & Sainitier, N. (2022). Effect of the geometry of butt-joint implant-supported restorations on the fatigue life of prosthetic screws. *Journal of Prosthetic Dentistry*, *127*(3), 477.e1-477.e9. https://doi.org/10.1016/j.prosdent.2021.12.010

Ayres, G., Macedo, A. P., Kubata, B. R., & Muglia, V. A. (2025). Effect of solid abutment diameter and implant placement depth on stress distribution in the posterior mandible: A finite element analysis study. *Computers in Biology and Medicine*, *188*(February), 109911. https://doi.org/10.1016/j.compbiomed.2025.109911

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

Cervino, G., Romeo, U., Lauritano, F., Bramanti, E., Fiorillo, L., D'Amico, C., Milone, D., Laino, L., Campolongo, F., Rapisarda, S., & Cicciù, M. (2018). Fem and Von Mises Analysis of OSSTEM ® Dental Implant Structural Components: Evaluation of Different Direction Dynamic Loads. *The Open Dentistry Journal*, *12*(1), 219–229. https://doi.org/10.2174/1874210601812010219

Chandrupatla, T. R. (1991). Introduction to finite elements in engineering. In *Choice Reviews Online* (Vol. 29, Issue 01). https://doi.org/10.5860/choice.29-0321

Cicciù, M., Cervino, G., Bramanti, E., Lauritano, F., Gudice, G. Lo, Scappaticci, L., Rapparini, A., Guglielmino, E., & Risitano, G. (2015). FEM analysis of mandibular prosthetic overdenture supported by dental implants: Evaluation of different retention methods. *Computational and Mathematical Methods in Medicine*, *2015*. https://doi.org/10.1155/2015/943839

Daniel, 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*, *12*(July), 1–7. https://doi.org/10.3389/fbioe.2024.1400918

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

Hibbeler, R. C. (2018). *Mechanics of Materials Tenth Edition in SI Units*.

Kinra, P., Jayakumar, K., Soumithran, C., Michael, M., Passi, D., & Singh, M. (2017). Comparative evaluation of bite force analytical study following mandibular osteosysthesis using three-dimensional and conventional locking miniplates. *National Journal of Maxillofacial Surgery*, *8*(1), 34. https://doi.org/10.4103/njms.njms_76_16

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

Moreno, A. L. D. M., Dos Santos, D. M., Bertoz, A. P. D. M., & Goiato, M. C. (2023). Abutment on Titanium-Base Hybrid Implant: A Literature Review. *European Journal of Dentistry*, *17*(2), 261–269. https://doi.org/10.1055/s-0042-1750801

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

Pjetursson, B., Asgeirsson, A., Zwahlen, M., & Sailer, I. (2014). Improvements in Implant Dentistry over the Last Decade: Comparison of Survival and Complication Rates in Older and Newer Publications. *The International Journal of Oral & Maxillofacial Implants*, *29*(Supplement), 308–324. https://doi.org/10.11607/jomi.2014suppl.g5.2

Quiudini, P. R., Pozza, D. H., Pinto, A. dos S., de Arruda, M. F., & Guimarães, A. S. (2017). Differences in bite force between dolichofacial and brachyfacial individuals: Side of mastication, gender, weight and height. *Journal of Prosthodontic Research*, *61*(3), 283–289. https://doi.org/10.1016/j.jpor.2016.10.003

Reif, T. J., Jacobs, D., Fragomen, A. T., & Rozbruch, S. R. (2022). Osseointegration Amputation Reconstruction. *Current Physical Medicine and Rehabilitation Reports*, *10*(2), 61–70. https://doi.org/10.1007/s40141-022-00344-9

Riset Kesehatan Dasar (Riskesdas). (2018). Laporan Riskesdas 2018 Nasional.pdf. In *Lembaga Penerbit Balitbangkes* (p. hal 156). https://repository.badankebijakan.kemkes.go.id/id/eprint/3514/1/Laporan Riskesdas 2018 Nasional.pdf

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). https://doi.org/10.1186/s40902-023-00401-w

Sotova, C., Yanushevich, O., Kriheli, N., Grigoriev, S., Evdokimov, V., Kramar, O., Nozdrina, M., Peretyagin, N., Undritsova, N., Popelyshkin, E., & Peretyagin, P. (2023). Dental Implants: Modern Materials and Methods of Their Surface Modification. *Materials*, *16*(23). https://doi.org/10.3390/ma16237383

Stoykov, M., Milkov, M., & Peev, S. (2022). *International Bulletin of Otorhinolaryngology Biocompatibility of materials and products used in medicine and dental medicine*.

Tan, Q. C., Huang, J. F., Bai, H., Liu, Z. X., Huang, X. Y., Zhao, X., Yang, Z., Du, C. F., Lei, W., & Wu, Z. X. (2022). Effects of Revision Rod Position on Spinal Construct Stability in Lumbar Revision Surgery: A Finite Element Study. *Frontiers in Bioengineering and Biotechnology*, *9*(January), 1–11. https://doi.org/10.3389/fbioe.2021.799727

Tribst, J. P. M., Dal Piva, A. M. de O., & Borges, A. L. S. (2016). Biomechanical tools to study dental implants: A literature review. *Brazilian Dental Science*, *19*(4), 5–11. https://doi.org/10.14295/bds.2016.v19i4.

1321

Verma, A., Jain, A., Sekhar Sethy, S., Verma, V., Goyal, N., Vathulya, M., & Kandwal, P. (2024). Finite element analysis and its application in Orthopaedics: A narrative review. *Journal of Clinical Orthopaedics and Trauma*, *58*(May). https://doi.org/10.1016/j.jcot.2024.102803

Yang, Y., Liu, Y., Yuan, X., Ren, M., Chen, X., Luo, L., Zheng, L., & Liu, Y. (2023). Three-dimensional finite element analysis of stress distribution on short implants with different bone conditions and osseointegration rates. *BMC Oral Health*, *23*(1), 1–14. https://doi.org/10.1186/s12903-023-02945-9

You, D., Mittal, R., Wang, M., & Moin, P. (2006). Analysis of stability and accuracy of finite-difference schemes on a skewed mesh. *Journal of Computational Physics*, *213*(1), 184–204. https://doi.org/10.1016/j.jcp.2005.08.007

Zainal Abidin, N. A., Ramlee, M. H., Ab Rashid, A. M., Ng, B. W., Gan, H. S., & Abdul Kadir, M. R. (2022). Biomechanical effects of cross-pin's diameter in reconstruction of anterior cruciate ligament – A specific case study via finite element analysis. *Injury*, *53*(7), 2424–2436. https://doi.org/10.1016/j.injury.2022.05.021