Stress interaction between mandibular bone and implant with maximum diameter using FEA

Wan Muhamad Amrizam Wan Mohd Azmi [1], Nurul Najwa Mansor[1], Mohd Sabri Hussin[1]

Faculty of Mechanical Engineering & Technology, Universiti Malaysia Perlis (UniMAP),02600, Arau, Perlis, Malaysia

Abstract. The purpose of this study is to identify the von misses stress distribution on trabecular bone and cortical bone at mandibular bone after react with single prostheses supported implants using the Finite Element Analysis (FEA) method. This analysis was carried out by producing 6.5mm diameter implant models, followed by trabecular bone and cortical bone models using Catia V5R21. After the model is produced FEA will be carried out using Abaqus CAE. Criteria for data collection is using Von Misses values. The result reaction from compression 200 N force applied is the stress for implant is accepted. While out of range for trabecular bone and cortical bone. In conclusion model improvements need to be done so that the obtained values are safe.

Keywords: Biomedical, Mandibular bone, Implant, Finite element analysis, Von misses stress.

1 Introduction

Implant diameter and bone quality are two major factors that influence the biomechanics of an implant-supported prosthesis [1]. The size of the dental implants plays a crucial role in distributing stress within the implants [2]. The size of dental implants plays an important role in the stress distribution in implants [2]. By controlling these elements, mechanical failures such as breakage of screws, components or facets on the framework can be avoided [3].

The larger diameter offers greater primary stability, especially in areas with dense bone, such as the posterior mandible [4]. Using an implant with maximum diameter from [5] Straumann Group, 6.5mm diameter have benne choose for analysis.

Length 10mm choose because Horiuchi et al. suggested that implants should be at least 10 mm long to ensure a high success rate[6]. Research indicates that 10mm-long implants have excellent success rates, especially in the back of the mouth where there may be limited bone height but increased chewing forces [4].

However, limited studies have been specially planned in this position. Reaction of implant diameter on bone stress distribution and implant stability in this region remain unclear. Therefore, it is necessary to understand the role of small or minimum implant diameter [6].

2 Materials & method

Figure 1 below show the example of analysis from previous research will use in this research simulation. Software Catia V5r21 was used to create the model. While the analysis for FEA software Abaqus were used. The institutional ethical committee (IMU R 216-2018) approved this FEA study. A cortical bone and trabecular bone in a mandibular bone was modelled in a rectangle shape to illustrate the bone structure using CATIA V5r21 as show in Figure 2 and Figure 3. Subsequently, the stress distribution of von Mises was analysed using ABAQUS. The meshing process was automatically conducted and detailed in Table 1 below.

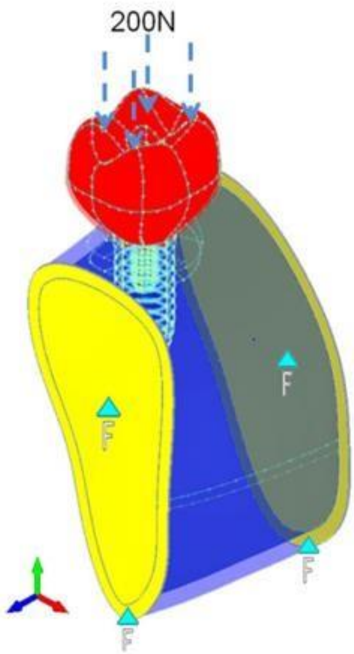

Figure 1: Example of analysis [5].

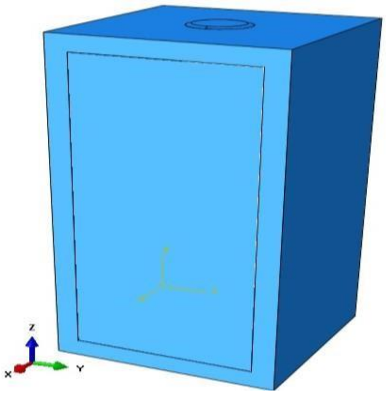

Figure 2: Model of analysis (Shaded view)

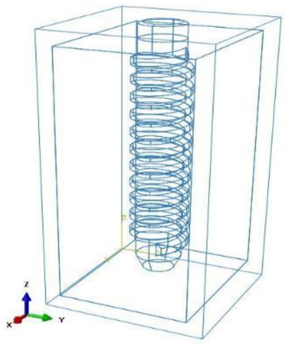

Figure 3: Model of analysis (wireframe view)

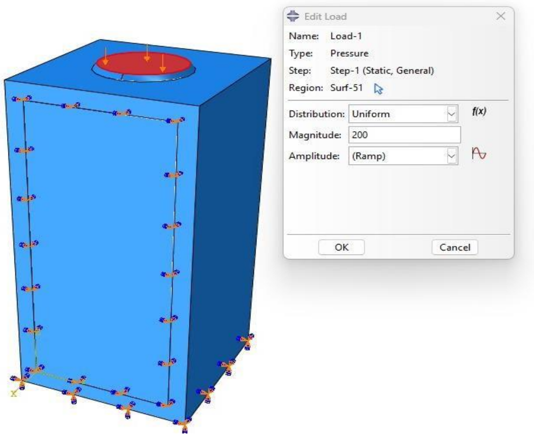

Figure 4: Load application and boundary condition.

From Figure 1, Using a 200N load in dental implant analysis is based on its relevance to simulating the forces typically experienced by implants during mastication. Studies show that this force is within the range commonly encountered during chewing and other oral functions, providing a realistic scenario for analysing implant performance and stress distribution. The application of 200N allows researchers to evaluate the biomechanical stability and stress response of implants under conditions that mimic everyday use. By applying this load, analysis can provide insights into how different implants respond to forces that they would regularly encounter, ensuring that the outcomes are clinically relevant [8].

From Figure 2 & 3, this analysis was designed like previous studies. however, each input used is a mixture of various sources. The choice was made to use a standard 3D model mini screw implant with a 6.5mm diameter, which corresponds to the maximum size in the standard screw implant (6.5 × 10mm) as described in Table 1 [9]. The chosen implant was designed as a single body using computer-aided design (CAD) software program CATIA V5r21 and then imported to ABAQUS to analyze its effect on the cortical and trabecular bone in the mandibular structure. Individual 3D models were put together with an intersecting contact-mesh with the bone surface. Figure 4, explain force direction applied at the top of implant model with 200N force. The material properties are detailed in Table 2.

Table 1: Implant size [9].

Size	Diameter (mm)	Length (mm)
1	6.5	10

Table 2: Implant materials and properties.

Materials	Young modulus (GPa)	Poisson Ratio (V)	Reference
Titanium	110	0.35	[2,7–9]
Trabecular bone	1.37	0.3	[2,8,10,11]
Mandibular bone	13.7	0.3	[2,7–9]

3 Result

The overall model assembly was indicated by using Von misses in any individual component, as shown in the Table 3 and Figure 5 until Figure 8 is simulation result

Table 3: Von misses result

Part	Von misses (MPa)
Implant	602.58
Trabecular bone	147.21
Cortical bone	113.71

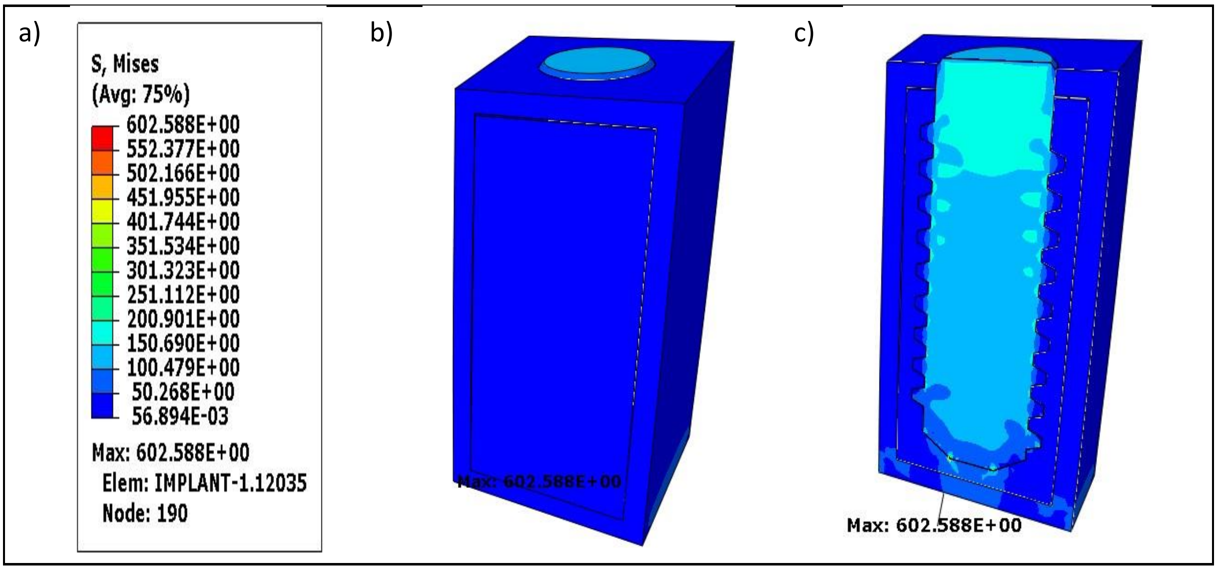

Figure 5: a) Contour plot Von misses Stress for model (implant ⌀ 6.5mm), b) Result analysis for model (implant ⌀ 6.5mm), c) Result analysis for model (implant ⌀ 6.5mm), (Cut Section).

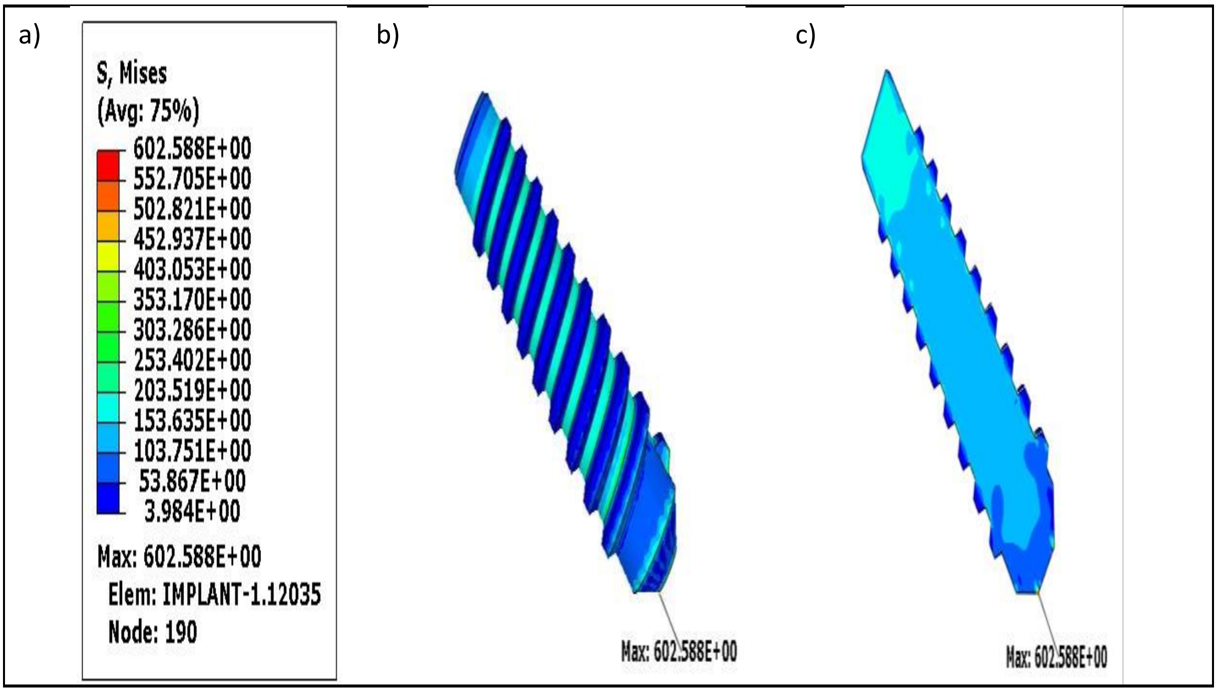

Figure 6: a) Contour plot Von misses Stress for implant (implant ⌀ 6.5mm), b) Result analysis for implant (implant ⌀ 6.5mm), c) Result analysis for implant (implant ⌀ 6.5mm), (Cut Section).

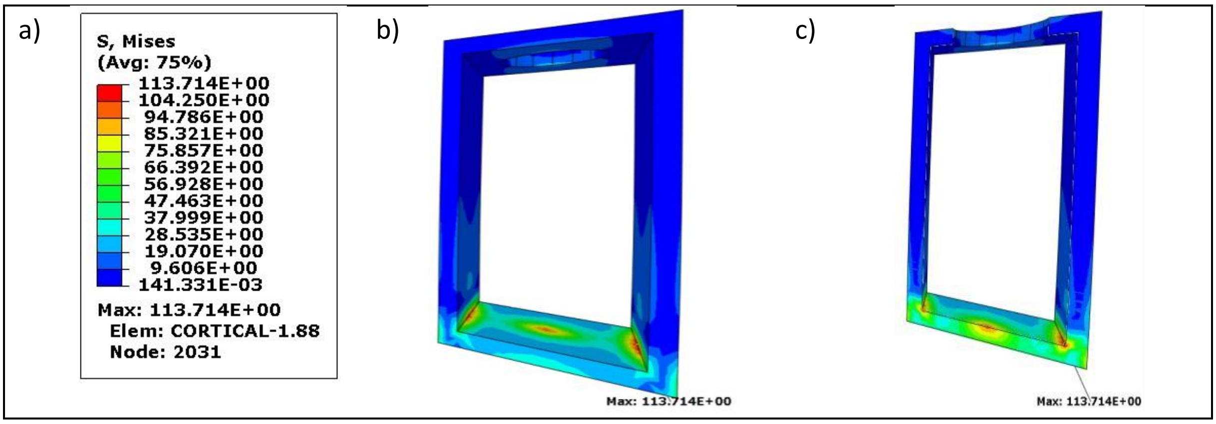

Figure 7: a) Contour plot Von misses Stress for Cortical bone (implant ⌀ 6.5mm), b) Result analysis for Cortical bone (implant ⌀ 6.5mm), c) Result analysis Cortical bone (implant ⌀ 6.5mm), (Cut Section).

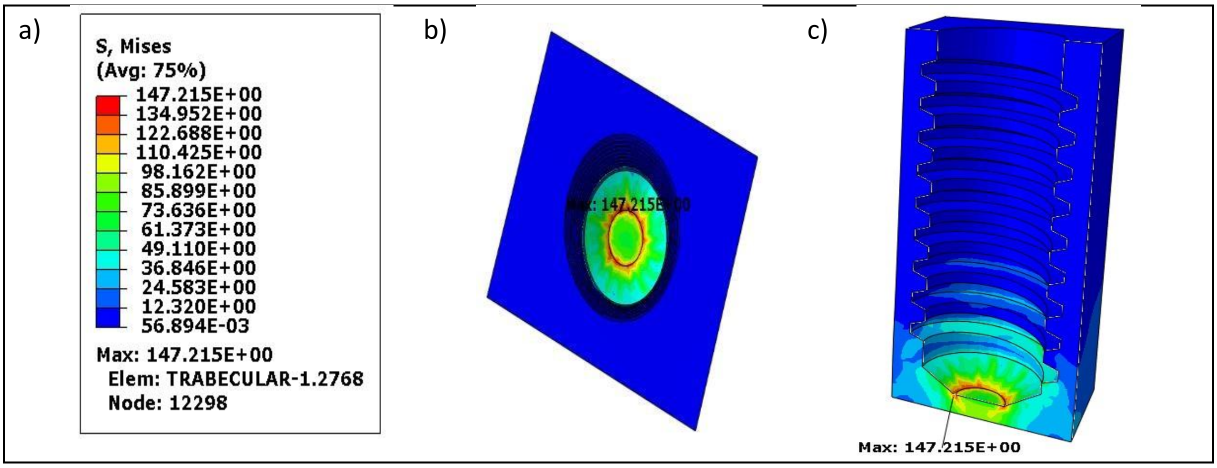

Figure 8: a) Contour plot Von misses Stress for Trabecular bone (implant ⌀ 6.5mm), b) Result analysis for Trabecular bone (implant ⌀ 6.5mm), (top view), c) Result analysis Trabecular bone (implant ⌀6.5mm), (Cut Section).

4 Discussion

From previous research, Table 4 is a range of Von misses stress should be achieved and cooperation with analysis result. The stress distribution is significantly affected by the implant's material properties, such as elasticity and strength. Implants with larger diameters generally provide a greater surface area for load distribution. However, they may not always reduce stress if other factors, such as implant length or thread design, are not optimized [12].

Table 4: Comparison between analysis result and Von misses stress range.

Part	Range (MPa)	Reference	Analysis result (MPa)	conclusion
Implant	50 – 800	[8]	602.58	In range
Trabecular bone	0.1 - 5	[13]	147.21	Out of range
Cortical bone	10 -50	[14]	113.71	Out of range

The Von Mises stress value for the implant is 602.58 MPa, quite high but still within the acceptable range for mandibular implants, depending on the implant material used. Implants have to withstand high loads especially in the mandibular area, where pressure is very large. Stress on implants can reach high values depending on the design and the applied load [13]. The high Von Mises stress value (147.21 MPa) on trabecular bone shows that this bone is under much higher pressure than normal. Trabecular bone usually has lower strength than cortical bone. This value may indicate a situation where the implant transmits a large load directly to the trabecular bone, which can lead to a risk of injury or fracture. Trabecular bone usually experiences lower stress, around 0.1 to 5 MPa. The value obtained is much higher, possibly due to factors such as implant design or certain anatomical conditions [14].

The Von Mises stress value obtained for cortical bone is 113.71 MPa. This is within the more normal range for mandibular cortical bone, where stress is typically higher than trabecular bone but lower than implant. Cortical bone acts as the main support structure that transfers the load from the implant to the rest of the mandible. This value is normal and shows the function of the cortical bone in channelling the load from the implant [15].

Clinicians must consider many factors when selecting dental implants. Several studies have analysed implant complexes through finite element modelling and loading tests, providing new [1].

The success of dental implantation depends on various factors, including implant diameter, implant length, bone quality, and other design factors such as thread features, implant system, and abutment collar height. Improving bone quality reduces bone strain values, and implants with 10 to 20-degree neck configurations are recommended to reduce strain values and enhance load dissipation in bone [1,16].

5 Conclusion

The study's conclusion is that the value obtained is appropriate. To ensure that the values obtained are safe, however, model improvements must be made. This study needs to be validated with clinical to ensure the results obtained are accurate. If this stress is found to be too high in a clinical context, there needs to be an adjustment in the design and simulation.

6 Acknowledgement

The author expresses many thanks to those who helped, especially the lecturers of the Faculty of Mechanical Engineering & Technology. Not forgetting financial support from (FGRS/1/2020/TK 0 UNIMAP/02/111)

7 References

1. Anitua E, Larrazabal Saez de Ibarra N, Saracho Rotaeche L. Implant-Supported Prostheses in the Edentulous Mandible: Biomechanical Analysis of Different Implant Configurations via Finite Element Analysis. Dent J (Basel). 2023 Jan 1;11(1).

2. Santiago Junior JF, Pellizzer EP, Verri FR, De Carvalho PSP. Stress analysis in bone tissue around single implants with different diameters and veneering materials: A 3-D finite element study. Materials Science and Engineering C. 2013 Dec 1;33(8):4700–14.

3. Lin D, Li Q, Li W, Swain M. Dental implant induced bone remodeling and associated algorithms. Vol. 2, Journal of the Mechanical Behavior of Biomedical Materials. 2009. p. 410–32.

4. Rameh S, Menhall A, Younes R. Key factors influencing short implant success. Oral Maxillofac Surg [Internet]. 2020;24(3):263–75. Available from: https://doi.org/10.1007/s10006-020-00841-y

5. Straumann. Straumann® BLX Implant System [Internet]. [cited 2024 Apr 4]. Available from: https://www.straumann.com/en/dental-professionals/products-and-solutions/dental-implants/blx.html

6. Li T, Hu K, Cheng L, Ding Y, Ding Y, Shao J, et al. Optimum selection of the dental implant diameter and length in the posterior mandible with poor bone quality - A 3D finite element analysis. Appl Math Model. 2011 Jan;35(1):446–56.

7. Darvish D, Khorramymehr S, Nikkhoo M. Finite Element Analysis of the Effect of Dental Implants on Jaw Bone under Mechanical and Thermal Loading Conditions. Math Probl Eng. 2021;2021.

8. Menacho-Mendoza E, Cedamanos-Cuenca R, Díaz-Suyo A. Stress analysis and factor of safety in three dental implant systems by finite element analysis. Saudi Dental Journal. 2022 Nov 1;34(7):579–84.

9. Xie B, Chen J, Zhao T, Shen J, Dörsam I, He Y. Three-dimensional finite element analysis of anterior fixed partial denture supported by implants with different materials. Annals of Anatomy. 2022 Aug 1;243.

10. Ye Z, Ye H, Jiang Z, Wu S, Feng Z, Liu H, et al. Influence of different implant designs on replacement of four teeth of the posterior free-end edentulism: Three-dimensional finite element analysis and clinic case validation. Annals of Anatomy. 2023 Aug 1;249.

11. Tomás M, Radaelli B, Idogava HT, Spazzin AO, Noritomi PY, Boscato N. Parafunctional loading and occlusal device on stress distribution around implants: A 3D finite element analysis. 2018.

12. Baggi L, Cappelloni I, Maceri F, Vairo G. Stress-based performance evaluation of osseointegrated dental implants by finite-element simulation. Simul Model Pract Theory. 2008 Sep;16(8):971–87.

13. Shinya A, Ishida Y, Miura D, Shinya A. The effect of implant length and diameter on stress distribution around single implant placement in 3d posterior mandibular fe model directly constructed form in vivo ct. Materials. 2021 Dec 1;14(23).

14. Zupancic Cepic L, Frank M, Reisinger A, Pahr D, Zechner W, Schedle A. Biomechanical finite element analysis of short-implant-supported, 3-unit, fixed CAD/CAM prostheses in the posterior mandible. Int J Implant Dent. 2022 Feb 11;8(1).

15. Lee H, Jo M, Noh G. Biomechanical effects of dental implant diameter, connection type, and bone density on microgap formation and fatigue failure: A finite element analysis. Comput Methods Programs Biomed. 2021 Mar 1;200.

16. Ausiello P, Tribst JPM, Ventre M, Salvati E, di Lauro AE, Martorelli M, et al. The role of cortical zone level and prosthetic platform angle in dental implant mechanical response: A 3D finite element analysis. Dental Materials. 2021 Nov 1;37(11):1688–97.