Effect on stress distribution with minimum implant diameter towards mandibular bone using FEA

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

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

Abstract. The purpose of this study is to evaluate the stress on trabecular bone and cortical bone located in trabecular bone with single prostheses supported by implants using the Finite Element Analysis (FEA) method. This experimental method was carried out by producing 3.5mm diameter implant models, followed by trabecular bone and mandibular 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 and cortical bone is accepted. While out of range for trabecular 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

The success of dental implant is related to the quality and quantity of jaw bones, implant design, implant surface texture. surgical procedures and so on. Among the implant designs, implant diameter intensively studied and well accepted as key factors, since they directly influence the primary stability, placement, and removal torque values of dental implant [1].

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].

Implants with a diameter of 3.5mm are often chosen for cases with narrow bony ridges or limited interdental space [4]. Smaller diameter than 3.5 mm are termed narrow or small-diameter implants [5]. Mini-implants carry mechanical risks including fracture of implant fixture and reduction in resistance to the occlusal force due to their smaller-diameter [5]. Previous studies have reported the potential risks that might occur as the diameter of implant lessens [5]. The implant design for this analysis are refer from [6].

Length 10mm choose because Horiuchi et al. suggested that implants should be at least 10 mm long to ensure a high success rate[1]. 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 [7].

However, few studies have been specially designed in this position. Effects 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 [1].

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.

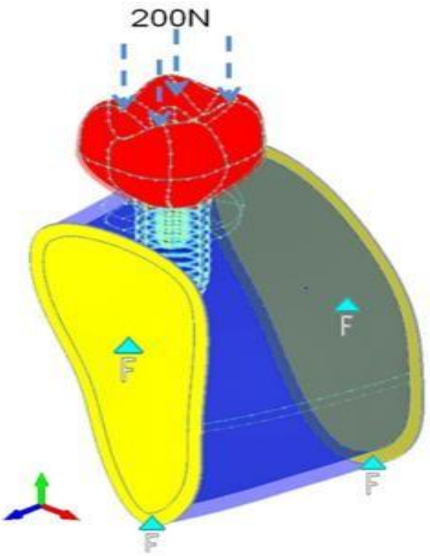

Figure 1: Example of analysis [2].

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.

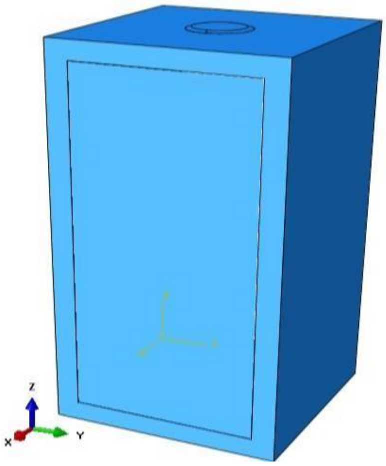

Figure 2: Model of analysis (Shaded view)

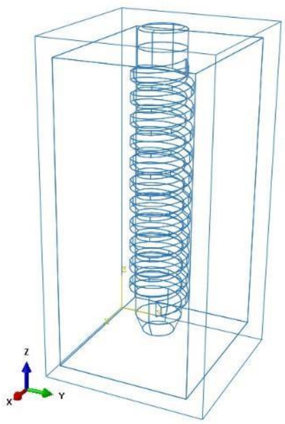

Figure 3: Model of analysis (wireframe view)

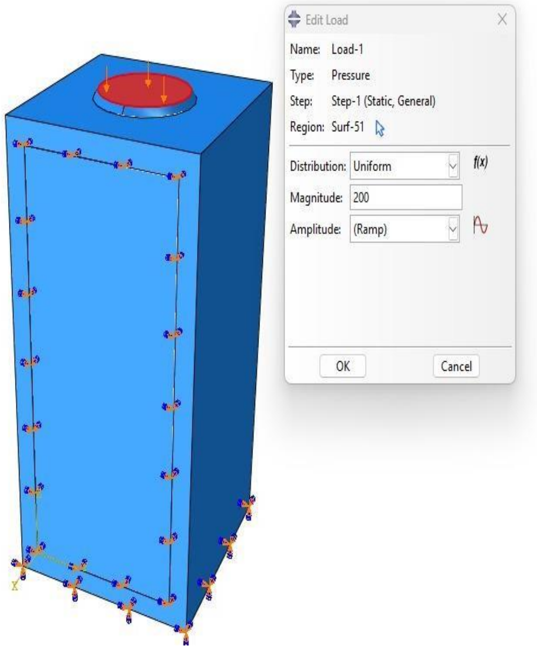

Figure 4: Load application and boundary condition.

From Figure 1, The use of a 200N load in dental implant analysis is chosen because it accurately represents the forces experienced by implants during chewing and other oral activities. Research indicates that this force level is commonly encountered during various oral functions, making it a practical method for examining implant performance and stress distribution. Employing a 200N load enables researchers to assess the biomechanical stability and stress reactions of implants in conditions that closely resemble everyday use. This approach allows for insights into how different implants react to the typical forces they face, ensuring that the findings have direct clinical relevance [9].

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 3.5mm diameter, which corresponds to the maximum size in the standard screw implant (Ø3.5 × 10mm) as described in Table 1 [[10]. The chosen implant was designed as a single body using computeraided design (CAD) software program CATIA V5r21 and then imported to ABAQUS to analyze its effect on the cortical and spongy 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 [10].

Size	Diameter (mm)	Length (mm)
1	3.5	10

Table 2: Implant materials and properties.

Materials	Young modulus (GPa)	Poisson Ratio (V)	Reference
Titanium	110	0.35	[5, 11–14]
Trabecular bone	1.37	0.3	[5, 14–16]
Mandibular bone	13.7	0.3	[5, 14–16]

3 Result

The overall model assembly was indicated by using Von misses stress 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	369.38
Trabecular bone	158.30
Cortical bone	37.61

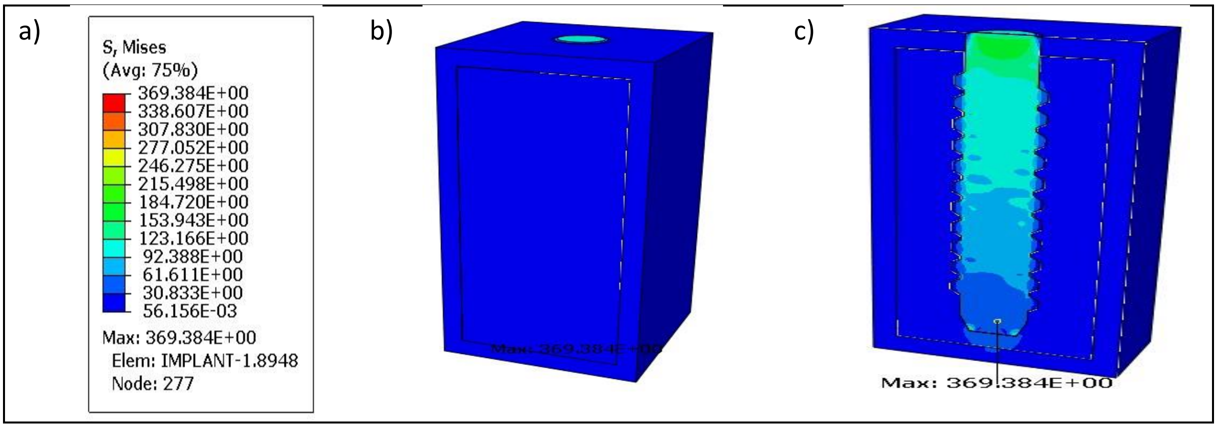

Figure 5: a) Contour plot Von misses Stress for model (implant ⌀ 3.5mm), b) Result analysis for model (implant ⌀ 3.5mm), c) Result analysis for model (implant ⌀ 3.5mm), (YZ-plane cross section).

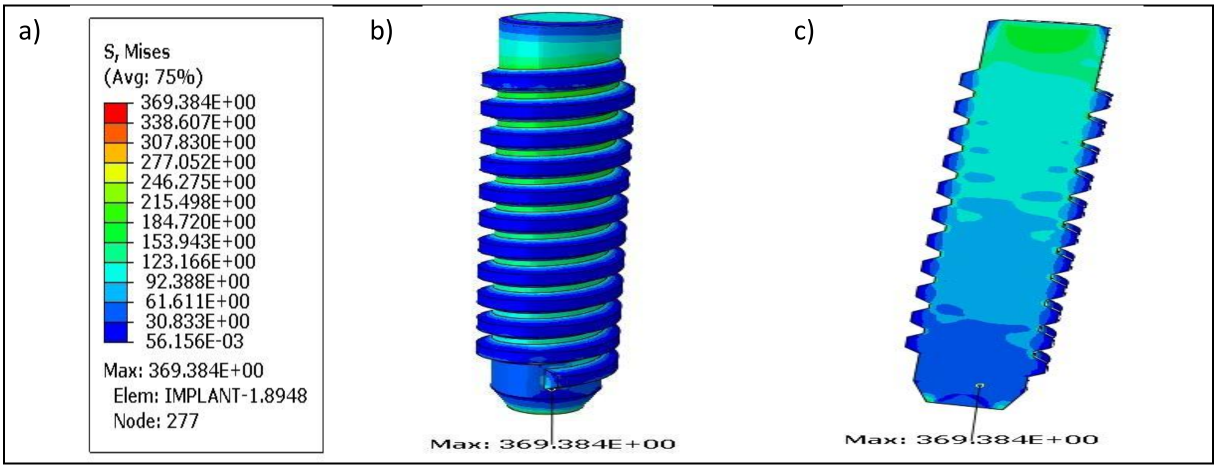

Figure 6: a) Contour plot Von misses Stress for implant (implant ⌀ 3.5mm), b) Result analysis for implant (implant ⌀ 3.5mm), c) Result analysis for implant (implant ⌀ 3.5mm), (YZ-plane cross section).

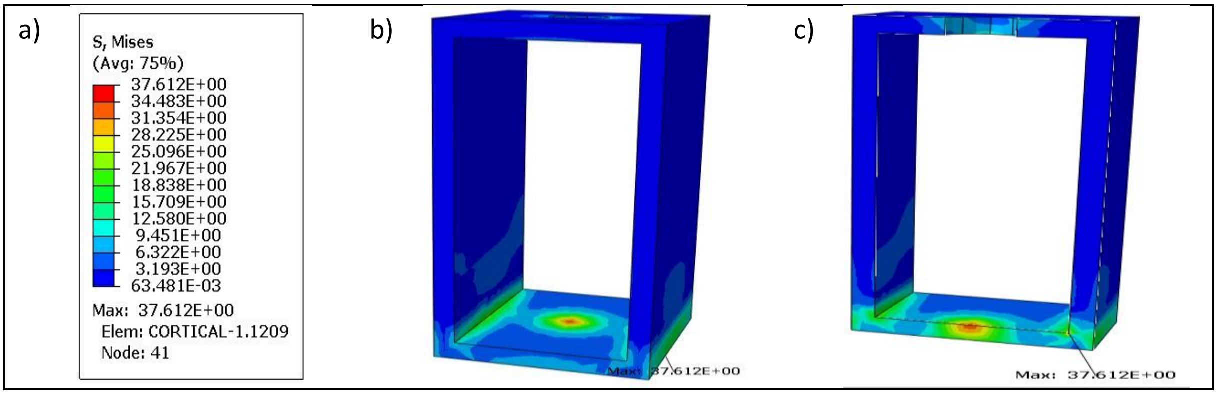

Figure 7: a) Contour plot Von misses Stress for Cortical bone (implant ⌀ 3.5mm), b) Result analysis for Cortical bone (implant ⌀ 3.5mm), c) Result analysis Cortical bone (implant ⌀ 3.5mm), (YZ-plane cross section).

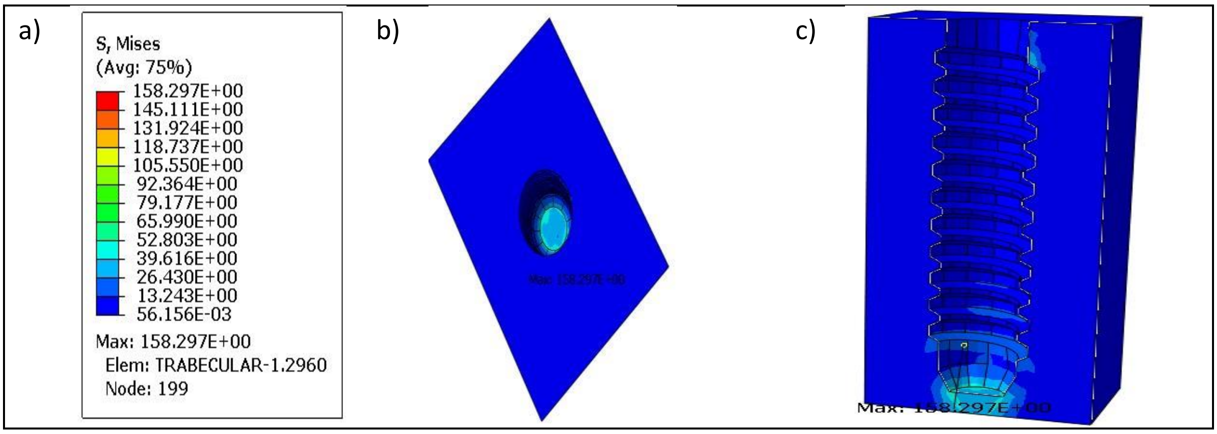

Figure 8: a) Contour plot Von misses Stress for Trabecular bone (implant ⌀ 3.5mm), b) Result analysis for Trabecular bone (implant ⌀ 3.5mm), (top view), c) Result analysis Trabecular bone (implant ⌀ 3.5mm), (YZ-plane cross section).

4 Discussion

Table 4 presents a range of Von misses stress that should be achieved and cooperation with analysis result based on previous research. Depending on the implant material used, the Von Mises stress value for the implant is 369.38 MPa, which is quite high but still within the acceptable range for mandibular implants. High loads must be tolerated by implants, particularly in the mandibular region where high pressure is present. Depending on the design and the applied load, implants may experience high levels of stress [8].

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

Part	Von misses stress range (MPa)	Reference	Analysis result (MPa)	conclusion
Implant	50 – 800	[14]	369.38	In range
Trabecular bone	0.1 - 5	[18]	158.30	Out of range
Cortical bone	10 -50	[19]	37.61	In range

The fact that trabecular bone has a high Von Mises stress value (158.30 MPa) indicates that it is under significantly more pressure than usual. Compared to cortical bone, trabecular bone typically has less strength. This number could point to an instance where the implant transfers a significant load straight to the trabecular bone, increasing the possibility of harm or fracture. Trabecular bone typically undergoes 0.1 to 5 MPa of stress. The value acquired is significantly higher, potentially as a result of elements like implant design or specific anatomical circumstances [9].

Cortical bone is generally stronger and able to withstand higher stress compared to trabecular bone. However, this value of 37.61 MPa is close to the stress limit that can cause damage, depending on the thickness of the cortical bone and other factors such as the age and bone health of the patient. It is important to ensure that the stress on the cortical bone remains within safe limits to avoid long-term problems. This may require further adjustments to implant design, load reduction, or optimization of implant placement [10].

When choosing dental implants, clinicians have a lot of factors to take into account. Numerous investigations have examined implant complexes using loading tests and finite element modelling, offering new [11]**.** Implant diameter, implant length, bone quality, and other design elements like thread characteristics, implant system, and abutment collar height all affect how well a dental implantation works. Implants with 10 to 20-degree neck configurations are advised to decrease strain values and improve load dissipation in bone because better bone quality lowers bone strain values [12,13].

5 Conclusion

Based on the study, the value obtained is acceptable. However, model improvements need to be done so that the obtained values are safe.

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. 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.

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. Heimes D, Becker P, Pabst A, Smeets R, Kraus A, Hartmann A, et al. How does dental implant macrogeometry affect primary implant stability? A narrative review. Int J Implant Dent. 2023 Jul 5;9(1).

5. Song SY, Lee JY, Shin SW. Effect of Implant Diameter on Fatigue Strength. Implant Dent. 2017 Feb 1;26(1):59–65.

6. 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

7. 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

8. 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.

9. El-Zawahry MM, Ibraheem EM, Zakaria Nassani M, Ghorab SA, El-Anwar MI. Stress analysis of mandibular implant overdentures retained with one, two, or four ball attachments: A finite element study [Internet]. Vol. 437, Dental Research Journal. 2018. Available from: www.ncbi.nlm.nih.gov/pmc/journals/1480

10. 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).

11. 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).

12. 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.

13. 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).