Engineering Technology
Open Access Journal

Research Article
Volume 3 Issue 3 - November 2019
DOI: 10.19080/ETOAJ.2019.03.555611

Eng Technol Open Acc

Selection of Optimum Thread Shape Design on Stress Distribution Among 5 Different Implant Types: A Finite Element Analysis

Yosef Wakjira, Song Jeng Huang* and Yi Hsiung Chiu

Department of Mechanical Engineering, National Taiwan University of Science and Technology, ROC

Submission: August 06, 2019; **Published:** November 15, 2019

***Corresponding author:** Song Jeng Huang, Department of Mechanical Engineering, National Taiwan University of Science and Technology, 43, Sec.4, Keelung Rd., Taipei, 106, Taiwan, ROC

Abstract

The choice of the thread structure of the implant plays a crucial role in the dental treatment. This study was undertaken to:

a) Identify the optimum design of the thread

b) Investigate the impacts of different implant thread models on bone implant-interface stress distribution

c) Investigate Static and other behaviors of implants

A full mechanical system design for a new dental implant model was developed within the range of the study. The design was optimized using the FEA depending on the design strategic objectives. Five implants with different thread profile designs considered and the pattern of the stress distribution in cortical bone and Cancellous bone are evaluated using FEA. The dimensions were in according with the commercial thread configuration standards for dental implant. The 3D CREO Parametric V5 software was used for implants as well as interface 3D modeling and Simulated in ABAQUS V2018 FEA Software. Applied load for both cases has a magnitude of 150N at 30o Angulation. Maximum stresses found concentrated at the cortical bone and transferred to the first thread of the implant. The Simulation shows that the least stresses were observed at the cancellous bone and maximum at the implant.

Keywords: Dental implant; Thread shape; Finite Element Analysis (FEA)

Introduction

Dental implant is an unnatural element that is used as a substitute for natural teeth once natural teethes removed in times such as accidental accidents, oral infection harm, birth and aging deformities. Threaded dentistry implants are currently widely used due to their elevated achievement levels [1]. The shape of implants was among the most controversial design aspects of the end osseous structures and could influence implant bio-mechanics [2]. The size of the implant affects the region of potential bone preservation. Factors such as occlusion, chewing strength, numbers of implants and implant positions within the prosthesis influence bone forces contiguous to the implant. Surface characteristics such as threads are built into the model to convert shear stresses into more durable strength kinds. This is why the majority of implant models are threaded because the thread form is especially crucial in altering the bone interface strength [3]. In optimization biomechanical behavior of implants, thread form and geometry are very essential.

Threads are used to achieve maximum preliminary contact, enhance the initial stability of the implant, expand the external area, and favor interfacial stress dissipation [4,5]. Dental thread design optimization can enhance clinical success. In the form of a thread for the dental implants, four kinds of threads were suggested: V-form, buttress, Reverse and square and Reverse buttress form threads. But the commons are limited to four excluding the reverse buttress thread. Each such profile works differently because of the distinction in the ability to pass the loads on it. The optimization of specific thread models must take place in accordance with the specified thread parameters if a new thread profile is to be created. The survival rates of the implant were recorded to improve with increasing diameters for the diameter left below the top threshold of 4.5mm. Implant size of more than 9.0mm has also been shown to decrease bone pressure and improve implant stabilization [6-8].

Saluja et al. [9], has studied the effects of length and diameter in Finite Element Analysis (FEA) on the stress distribution trend of INDIDENT dental implants. Anwar et al. [10], analyzed dental implant and surrounding bones load transfer. Their objective was to study the stress distribution and the movement of various threaded implant designs of different diameters and lengths. E Esmail et al. [11], used 3 dimensional finite element analysis to evaluate the influence and density of peri-implant tissue structure on the biomechanical behaviour.

Eraslan & Inan [12], Studied the impact of thread design on the stress distribution of a solid screw implant using Cosmos Works 3D FEA. Huang et al. [13], Use FEM to analyze the effects of dental implant surface roughness and thread length on bone stress and interfacial slides. They produced solid 3D models of four types of threads including the V-thread, the buttress thread, the reverse buttress thread and the square thread. Their research purpose is determining optimal thread design. The main purpose of this study was to in order to select best suitable optimum implant thread shape and type.

Dental Implant System

Two main parts comprise a standard implant scheme, an abutment and an implant. The implant is the fundamental element that is positioned in the jawbone surgically. It is fixed on the hole in the implant and often attached to the top of the crown. In addition to the implant design conditions and abutments, mechanical system design of the implant-abutment interface is essential regarding system mechanical integrity, mechanical load reactions within the system, system stability, strength and the dental implant long-term survival. The shape of the implant influences the stress distribution considerably, though not the maximum stress levels at the junction between bone and implant. Cylindrical design is preferable, as it creates a more consistent stress profile to less concentrated stress levels in the body relative with conical implants [14]. For poor bone performance areas, it can be changed to conical because conical internal structure leads to enhanced main stabilization by compressing the smooth tissue during and following implantation [15,16]. A threaded profile is used instead of a straight or stepped ground form because the pressure concentrations in the trabecular tissue have been checked to decrease [17].

Materials and Methods

CAD geometry

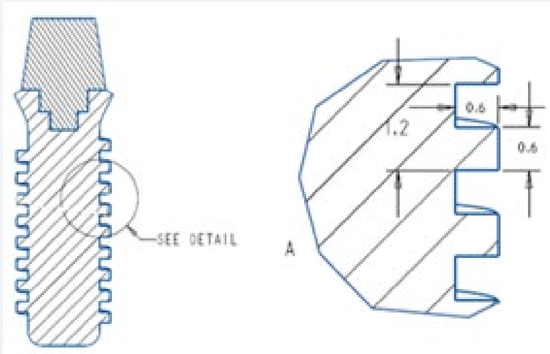

Figure 1: Implant Design and Parameters (Square Thread).

The 3D CREO Parametric V5 software was used for implants 3D modeling and ABAQUS V2018 FEA Software is used for simulation. Thread shape types that are used in the study: V-Thread, Buttress, Reverse buttress, Trapezoidal and Square thread profile. Length of the Implants selected is 12mm whereas the diameter of the Implants and the cortical bone thickness is 4mm and 2mm respectively. The outer dimension of the bone sections is modelled as 30mm height and 20mm width (Figure 1). Inside the cortical bone layer there is a cancellous bone layer modelled with 26mm height and 16mm width (Figure 2).

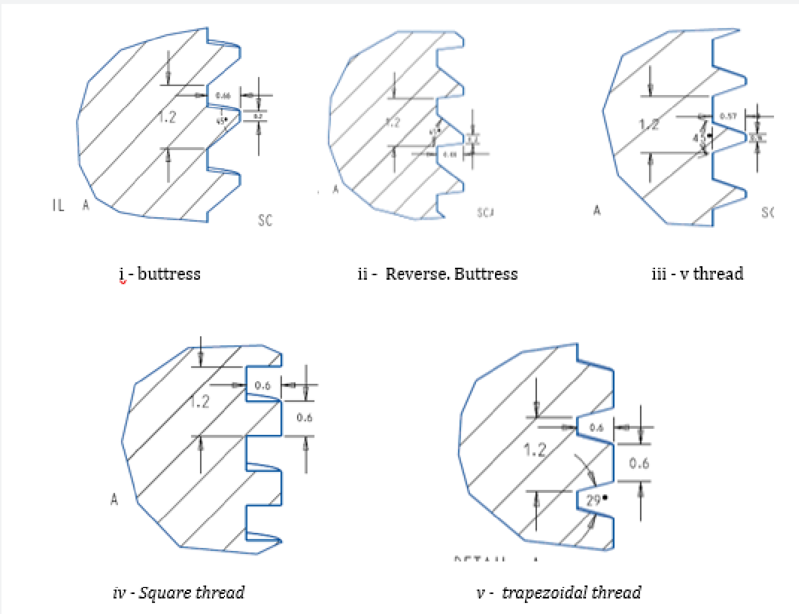

Figure 2: Schematic diagram of dimensions standardized for thread design.

How to cite this article: Yosef Wakjira, Song Jeng Huang, Yi Hsiung Chiu. Selection of Optimum Thread Shape Design on Stress Distribution Among 5 Different Implant Types: A Finite Element Analysis. Eng Technol Open Acc. 2019; 3(3): 555611. DOI: 10.19080/ETOAJ.2019.03.555611

FE modelling

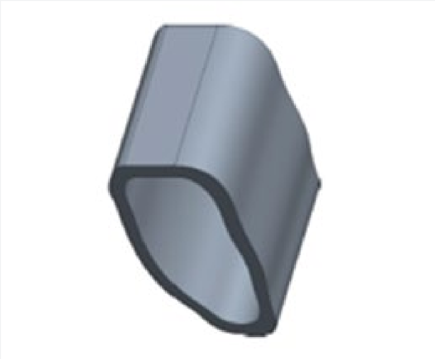

Figure 3: Bone Geometry Cortical bone.

Figure 3b: Assembled cortical.

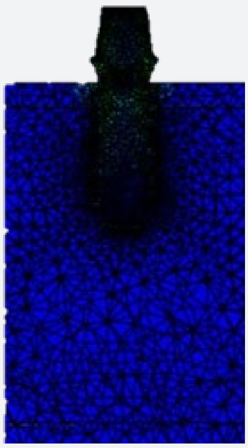

Figure 3c: cut section of Finite element model.

Using Hypermesh as a preprocessor the three-dimensional (3D) model of the implant and the implant system were meshed. Figure 3A-C shows the cut section of Finite element model. The interfaces of bone implant were assumed to have 100% Osseo integration. Cortical and cancellous bones sides and bottom were fully restricted and the limit conditions extended to the corresponding node. The implant from top to bottom was subjected to multi-constriction to limit root freedom. Static loading has been used to determine the bone model of the implant and the stress analysis interface. Von Mises stresses were the major factors used to measure levels of stress and evaluate the stresses of the implant and cortical bones.

Material property

Implant and abutment housing are relatively rigid components of the implant system proposed. Ti-6Al 4V a titanium alloy which has proven to be perfectly biocompatible and has good mechanical properties such as fracture strength, yield resistance and weariness, has been defined for these Implant components in terms of material. While Porcelain is selected for Abutment. It was therefore commonly used as a material of dental implant by commercial implant suppliers in biomedical implementations. It was assumed that the relatively rigid components have isotropic as well as linear elastic behavior containing small strains and small distortion. All bone features including implant, Abutment, cortical bone and cancellous bone are assumed to be homogeneous Anisotropic material. While Implant and abutment is assumed to be Isotropic material (Table 1 & 2).

Table 1: Isotropic Material Property for the Implant and Abutment [18].

	Youngs M (GPa)	Poisson's ratio
Porcelain	70	0.19
Ti-6Al-4V	110	0.35

Table 2: Anisotropic Material Property for the bone [19].

	Cancellous Bone (Gpa)	Cortical Bone (Gpa)
EX (MPa)	0.21	12.7
EY (MPa)	1.148	17.9
EZ (MPa)	1.148	22.8
GXY (MPa)	0.068	5
GYZ (MPa)	0.434	7.4
GXZ (MPa)	0.068	5.5
νXY	0.055	0.18
νYZ	0.322	0.28
νXZ	0.055	0.31

Bite force application

All dental implant system components were characterized as deformable contacts. No friction between contact bodies was defined as a simplification. A vertical and Axial force of 150N at an angle of 30 degree was applied to the load that controlled by the rigid structure that was connected to the top end of the abutment cap to simulate the chewing forces on both the implant-supported crown following use of forces and displacement constraints.

Boundary condition and load applied

A series of analyzes have been carried out after correctly defining the above mentioned parameters of analysis. The geometric pattern, material definitions and mesh creation on the

How to cite this article: Yosef Wakjira, Song Jeng Huang, Yi Hsiung Chiu. Selection of Optimum Thread Shape Design on Stress Distribution Among 5 Different Implant Types: A Finite Element Analysis. Eng Technol Open Acc. 2019; 3(3): 555611. DOI: 10.19080/ETOAJ.2019.03.555611

3D model were subsequently varied in the analyzes pertaining to an iterative procedure up to the desire results obtained.

Results and Discussion

The results are obtained based on Von Mises Stress distribution criterion. Maximum shear stress of 5 implants were obtained at different mesh size for each of the Implant.

Implant

(Figure 4)

Maximum Stress of Implants found on V Thread – 274.3Mpa (Figure 5).

Minimum stress of Implant found on Trapezoidal Thread – 66.08Mpa (Figure 6).

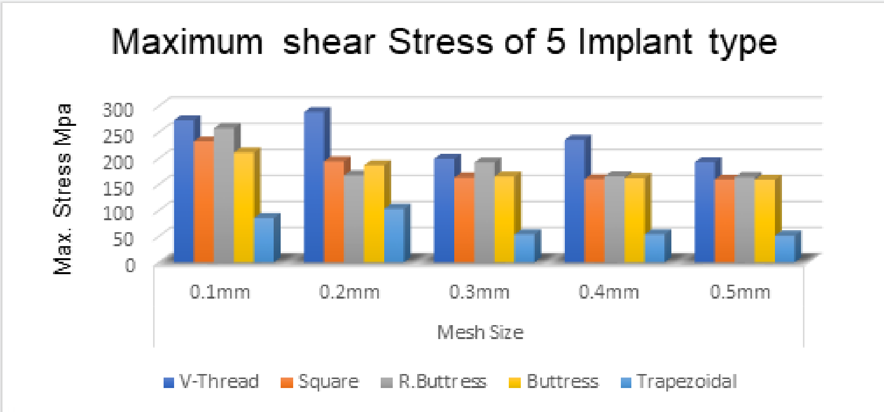

Figure 4: Shear stress of 5 Implant types at different mesh sizes.

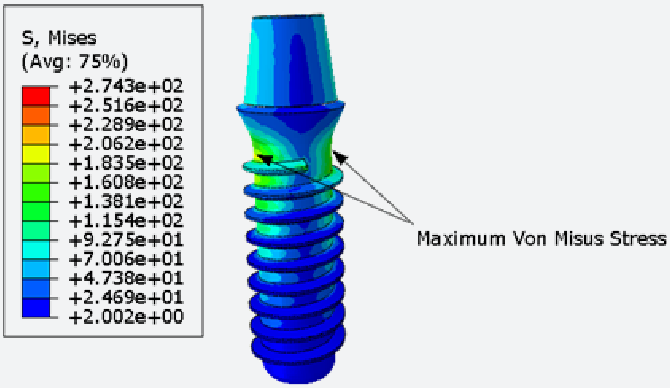

Figure 5: Maximum Von Mises at V thread design at the implant.

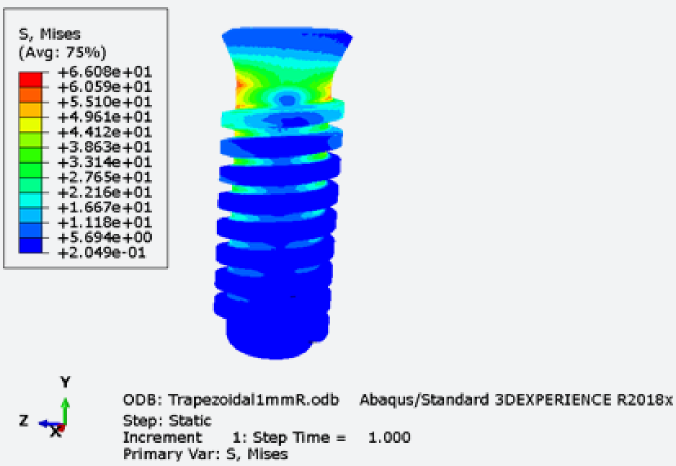

Figure 6: Maximum Von Mises at V thread design at the implant.

How to cite this article: Yosef Wakjira, Song Jeng Huang, Yi Hsiung Chiu. Selection of Optimum Thread Shape Design on Stress Distribution Among 5 Different Implant Types: A Finite Element Analysis. Eng Technol Open Acc. 2019; 3(3): 555611. DOI: 10.19080/ETOAJ.2019.03.555611

Cortical bone

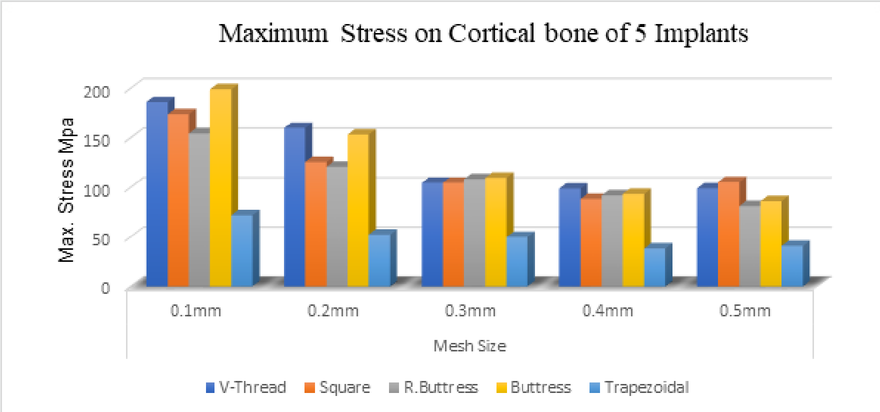

Figure 7: Max. Shear stress on Cortical bone of 5 Implant types at different mesh sizes.

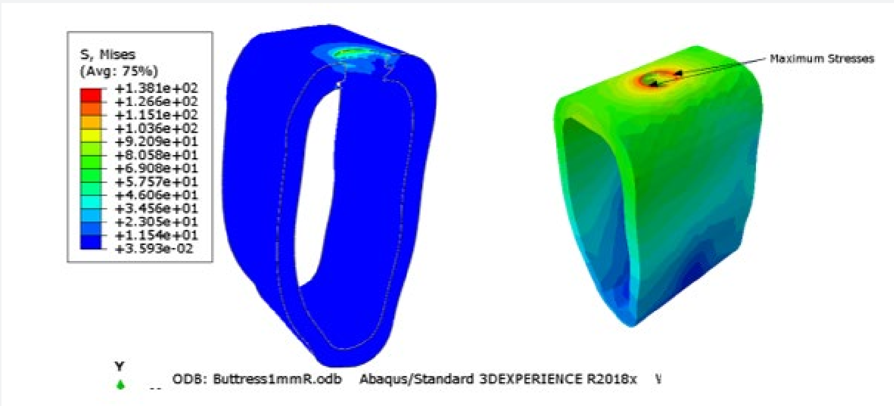

Figure 8: Von Mises stress at cortical bone of buttress thread implant.

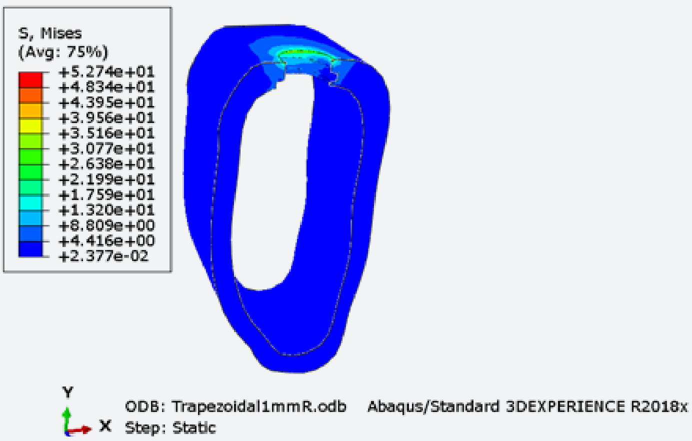

Figure 9: Von Mises stress of cortical bone on Trapezoidal thread implant.

(Figure 7)

Maximum Stress of Cortical bone found on Buttress Thread – 138.1Mpa (Figure 8).

Minimum stress of Cortical bone found on Trapezoidal Thread – 52.74Mpa (Figure 9).

Research by Mosavar reveals the supremacy of a square thread setup, as it showed the smallest stress in the implant cortical Bone Progress area of a square threads in osseointegration [20]. This research thus fails to take square thread into account and assesses V-film, buttress and inverse thread models. Misch claims three thread features which improve the operational surface, Increases

How to cite this article: Yosef Wakjira, Song Jeng Huang, Yi Hsiung Chiu. Selection of Optimum Thread Shape Design on Stress Distribution Among 5 Different Implant Types: A Finite Element Analysis. Eng Technol Open Acc. 2019; 3(3): 555611. DOI: 10.19080/ETOAJ.2019.03.555611

the initial contact and enable stress dissipation in the interface [21]. Interfacial stress examines demonstrated that threaded structures reduce stress close the thread area. In addition, other clinical benefits for threaded kinds, enhanced stabilization and stress-related tissue formation, can be considered.

Study carried out by Gonsalves was aimed at assessing the effect of an implant on the results of a 2D FEA with and without the thread [22]. The stress distribution throughout the model and within the cortical bone did not seem to have a strong impact on the implant, trabecular bone or screw. The maximum stress distribution of Von Mises on cortical bone-implant interfaces indicates no variations in either template, including trabecular bone. It has been shown that the distribution of stress throughout the cortical bone is not significantly affected by the shape of the implant, based on similar implant lengths and implant necks unlike the conclusions drawn in this study, which indicated variations in the thread design.

In this study Maximum Stress of Cortical bone found on Buttress Thread – 138.1Mpa while Minimum stress of Cortical bone found on Trapezoidal Thread – 52.74Mpa. So, for preservation of cortical bone what is needed is the stress to be reduced on bone, thus trapezoidal thread is preferable. In finding the optimum stress on a bone it is necessary to see the least stress magnitude.

Cancellous Bone

(Figure 10)

Maximum Stress of Spongy bone found on Buttress Thread – 41.37Mpa (Figure 11).

Minimum stress of Spongy bone found on Trapezoidal Thread – 10.27Mpa (Figure 12).

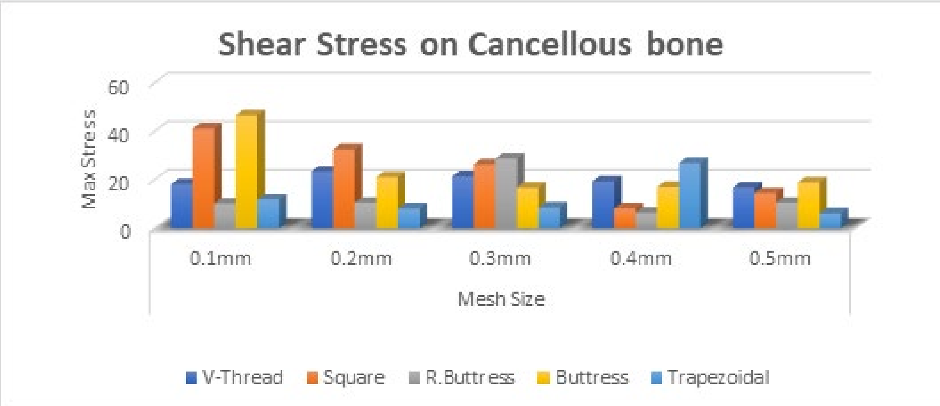

Figure 10: Maximum Shear stress on Cortical bone of 5 Implant types at different mesh sizes.

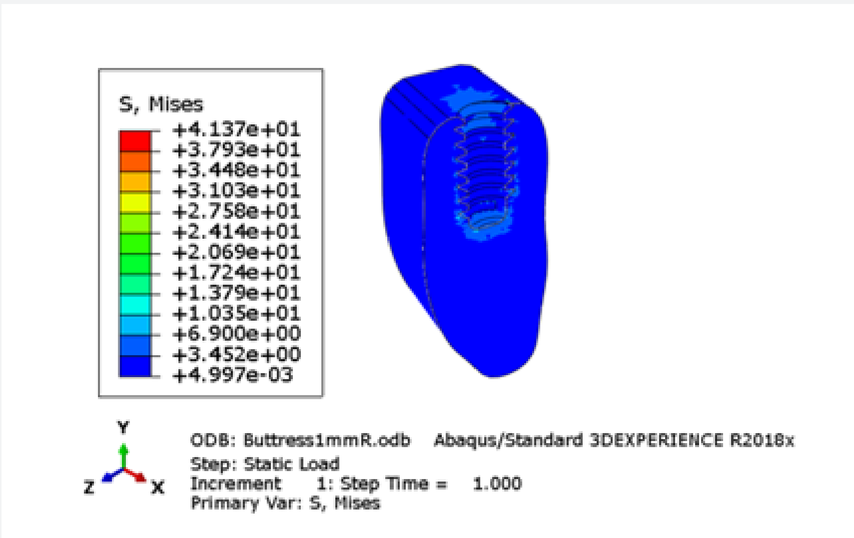

Figure 11: Von Mises stress of cancellous or spongy bone on Buttress threaded implant.

In a cancellous bone, Maximum Stress is found on Buttress Thread which is 41.37Mpa and Minimum stress found on Trapezoidal Thread – 10.27Mpa. Thus, again the same principle for cortical bone the least amount of stress magnitude is selected. Therefore, Trapezoidal thread again selected for optimizing the stresses on Trabecular bone.

How to cite this article: Yosef Wakjira, Song Jeng Huang, Yi Hsiung Chiu. Selection of Optimum Thread Shape Design on Stress Distribution Among 5 Different Implant Types: A Finite Element Analysis. Eng Technol Open Acc. 2019; 3(3): 555611. DOI: 10.19080/ETOAJ.2019.03.555611

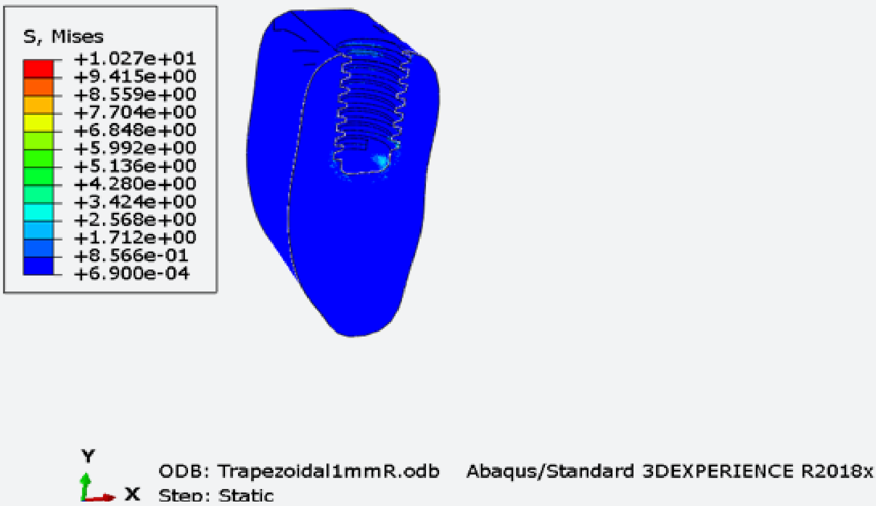

Figure 12: Von Mises stress of cancellous (spongy) bone on Trapezoidal thread implant.

Conclusion

1. Minimum stress from Mises has been concentrated by Trapezoidal, which is favorable for the preservation of the bone.

2. Trapezoidal Thread which is not introduced so much in dental implant type shows good performance with its large bone implant contact, stress distribution and bone preservation.

3. The maximum stress of implants and the minimum stress of the Trabecular bone were observed. The interface stress distribution depends heavily on the implant structure.

4. The Maximum stress on Implants found on V-Thread design. Whereas as the minimum for Implant exists on trapezoidal thread.

5. The cortical bone showed maximum stress in comparison to the Spongy bone in both cases for the Implant and for the Interface.

6. An optimum state of osseointegration between the cortical bone, cancellous bone and implant in the model was assumed which is not clinically present.

7. As the maximum stresses found around the neck of the Implant, The implant's neck needs to be strong enough. It may have an impact on implantation integrity if the Implant is not solid in this region.

References

1. Dhatrak P, Shirsat U, Sumanth S, Deshmukh V (2018) Finite element analysis and experimental investigations on stress distribution of dental implants around implant-bone interface. Materials Today: Proceedings 5(2): 5641-5648.
2. Lin S, Shi S, LeGeros RZ, LeGeros JP (2000) Three-dimensional finite element analyses of four designs of a high-strength silicon nitride implant. Implant Dent 9(1): 53-60.
3. Morris HF, Winkler S, Ochi S, Kanaan A (2001) A new implant designed to maximize contact with trabecular bone: Survival to 18 months. J Oral Implantol 27(4): 164-173.
4. Li T, Hu K, Cheng L, Ding Y, Ding Y, et al. (2011) Optimum selection of the dental implant diameter and length in the posterior mandible with poor bone quality–A 3D finite element analysis. Applied Mathematical Modelling 35(1): 446-456.
5. Geramizadeh M, Katoozian H, Amid R, Kadkhodazadeh MJ, Surgeons M (2018) Three-dimensional optimization and sensitivity analysis of dental implant thread parameters using finite element analysis. J Korean Assoc Oral Maxillofac Surg 44(2): 59-65.
6. Baggi L, Cappelloni I, Di Girolamo M, Maceri F, Vairo GJ (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. J Prosthet Dent 100(6): 422-431.
7. Kong L, Gu Z, Hu K, Zhou H, Liu Y, et al. (2009) Optimization of the implant diameter and length in type B/2 bone for improved biomechanical properties: A three-dimensional finite element analysis. Advances in Engineering Software 40(9): 935-940.
8. Li T, Kong L, Wang Y, Hu K, Song L, et al. (2009) Selection of optimal dental implant diameter and length in type IV bone: a three-dimensional finite element analysis. Int J Oral Maxillofac Surg 38(10): 1077-1083.
9. Saluja B, Alam M, Ravindranath T, Mubeen A, Adya N, et al. (2012) Effect of length and diameter on stress distribution pattern of INDIDENT dental implants by finite element analysis. Journal of Dental Implants 2(1): 19-25.
10. El-Anwar M, El-Zawahry M, El-Mofty M (2012) Load transfer on dental implants and surrounding bones. Aust J Basic Appl Sci 6(3): 551-560.
11. Esmail E, Hassan N, Kadah Y (2010) A three-dimensional finite element analysis of the osseointegration progression in the human mandible. In Medical Imaging 2010: Visualization, Image-Guided Procedures, and Modeling. 2010. International Society for Optics and Photonics.
12. Eraslan O, İnan Ö (2010) The effect of thread design on stress distribution in a solid screw implant: a 3D finite element analysis. Clin Oral Investig 14(4): 411-416.
13. Huang HL, Tsai MT, Hsu JT, Fuh LJ, Tu MG, et al. (2011) Do Threaded Size and Surface Roughness Affect the Bone Stress and Bone-Implant Interfacial Sliding of Titanium Dental Implant? In Proceedings of the World Congress on Engineering, London, England.

How to cite this article: Yosef Wakjira, Song Jeng Huang, Yi Hsiung Chiu. Selection of Optimum Thread Shape Design on Stress Distribution Among 5 Different Implant Types: A Finite Element Analysis. Eng Technol Open Acc. 2019; 3(3): 555611. DOI: 10.19080/ETOAJ.2019.03.555611

14. Mailath G, Stoiber B, Watzek G, Matejka M (1989) Bone resorption at the entry of osseointegrated implants--a biomechanical phenomenon. [Finite element study]. Z Stomatol 86(4): 207-216.

15. Åstrand P, Billström C, Feldmann H, Fischer K, Henricsson V, et al. (2003) Tapered Implants in Jaws with Soft Bone Quality: A Clinical and Radiographie 1-Year Study of the Brånemark System Mark IV Fixture. Clin Implant Dent Relat Res 5(4): 213-218.

16. Friberg B, Jisander S, Widmark G, Lundgren A, Ivanoff CJ, et al. (2003) One-year prospective three-center study comparing the outcome of a "soft bone implant"(prototype Mk IV) and the standard Brånemark implant. Clin Implant Dent Relat Res 5(2): 71-77.

17. Tada S, Stegaroiu R, Kitamura E, Miyakawa O, Kusakari HJ, et al. (2003) Influence of implant design and bone quality on stress/strain distribution in bone around implants: a 3-dimensional finite element analysis. Int J Oral Maxillofac Implants 18(3): 357-368.

18. Desai SR, Karthikeyan I, Gaddale R (2013) 3D finite element analysis of immediate loading of single wide versus double implants for replacing mandibular molar. J Indian Soc Periodontol 17(6): 777-783.

19. Rajendran SV, Shamnadh M, Dileep PJ (2014) Stress analysis of dental implants. 4(2): 115-117.

20. Rismanchian M, Birang R, Shahmoradi M, Talebi H, Zare RJ (2010) Developing a new dental implant design and comparing its biomechanical features with four designs. Dent Res J (Isfahan) 7(2): 70-75.

21. Lan TH, Du JK, Pan CY, Lee HE, Chung WH (2012) Biomechanical analysis of alveolar bone stress around implants with different thread designs and pitches in the mandibular molar area. Clin Oral Investig 16(2): 363-369.

22. Güngör H, Baran İ, Karaagaçlıoglu L (2018) Influence of Cantilever Extension and Crown-to-Implant Ratio in Posterior Maxilla: A Finite Element Analysis. Journal of Biomaterials and Tissue Engineering 8(4): 489-495.

How to cite this article: Yosef Wakjira, Song Jeng Huang, Yi Hsiung Chiu. Selection of Optimum Thread Shape Design on Stress Distribution Among 5 Different Implant Types: A Finite Element Analysis. Eng Technol Open Acc. 2019; 3(3): 555611. DOI: 10.19080/ETOAJ.2019.03.555611