ARPN Journal of Engineering and Applied Sciences

www.arpnjournals.com

EVALUATION OF STRESS DISTRIBUTION AND MICROMOTION OF DENTAL IMPLANT: IN VIVO CASE STUDY

Maya Genisa[1], Zainul Ahmad Rajion[1], Solehuddin Shuib[2], Dasmawati Mohamad[1] and Abdullah Pohchi[1]
[1]School of Dental Sciences, USM, Kubang Kerian, Malaysia
[2]Faculty of Mechanical Engineering, UITM, Johor Bahru, Malaysia
E-Mail: mgenisa@gmail.com

ABSTRACT

The cortical thickness is one of important factors for supporting the implant stability. There is a clear correlation between available cortical thicknesses for implant placement with implant stability. However, the relation between cortical thicknesses with stress distribution behaviour that might be generated during regular mastication process is still unclear. This study aimed to investigate the effect of cortical thickness on stress distribution of dental implant system by using in vivo model with finite element analysis (FEA). The in vivo model consisted of body implant, crown, two neighbour teeth, cortical and trabecular bone. The thicknesses of cortical thickness are varying: 2.30 mm, 2.85 mm, 3.53 mm, and 3.93 mm. The in vivo models are derived from Cone Beam Computed Tomography (CBCT) scanning with the segmentation process is performed by using MIMICS software with the pseudo cortical thickness are generated by using morphology tools which is available on this software. A 200 N vertical force are applied as an external loading into top of crown surface to simulate the masticatory process. The finite element analyses are performed in the ANSYS WORKBENCH 14 software and the maximum Von Mises Stress are selected to represent the stress distribution on each simulation. The result shows that the cortical thickness can protect the lowering bone area from overloading force in the post crown condition during masticatory. The stress will be propagated into cortical which is distributed to whole direction in the cortical and small part of stress are propagated to the trabecular bone surrounding implant. However, the thicker cortical thickness will transfer more stress into lower part of the jaw system particularly the trabecular bone.

Keywords: stress distribution, loading, implant, simulation, finite element analysis.

INTRODUCTION

The trend of dental implants and improvement of the methods were increased in the last decade [1]. Not only for improving appearance and speech, but also the dental implant can help for easier eating like natural teeth and it is a main objective of the dental implant placement. The success of dental implants for long lifetime depend on varying factors including how implant are placed, the foundation strength of the implant system and how the loading during mastication are distributed into the whole dental system especially in the early Osseo integration phases post-surgery.

Stress distribution around dental implant during loading is a main determinant for the success of osseointegration of dental implant after surgery. The stress distribution generated as a response to the external loading is dependent on how the load transferred into implants and it propagated into whole dental implant system. There are many factors affecting the stress distribution such as: the type of loading, the bone–implant interface, the length and diameter of the implants, the shape and characteristics of the implant surface, the prosthesis type, and the quantity and quality of the surrounding bone [2].

The objective of this paper is to investigate the effect of bone quantity (cortical thickness) on the stress distribution by using Finite element analysis (FEA) method for certain loading type (vertical loading) to simulate the masticatory process.

BIOMECHANICAL ASSESSMENT ON DENTAL IMPLANT SYSTEM

Bone modelling and remodelling after dental implant placement is an internal to add or remove the bone from periosteal and endosteal surface which is characterized by changes in bone density. When the external loadings are applied into dental system, bone responds by adjusting its mass density. If the mechanical loading (stress, strain, strain energy density and fatigue) conditions deviate from homeostatic levels, the stimulus to generate a new bone are decreased [3]. And hence the biomechanical assessment on dental implant system during loading can help us to understand the mechanism of osseo integration to support a long lifetime of the implant.

In theoretical method, the assessment of biomechanical of dental system can be done by using finite element analysis (FEA). FEA is defined as "a theoretical method that can be used for calculating the behavior of a real structure by performing algebraic solutions of a set of equations describing an idealized model structure with a finite number of variables" [4]. This method is a non-destructive technique for quantifying stresses and strain distribution generated by external loading [5]. The advantages of using FEA are this method is applicable for evaluating the complex geometry shapes which is difficult to be performed in the laboratory physically.

The stress distribution resulted from a simulation of FEA is depending on the nature of the mechanical applied loading. The generated stress, on the dental implant and also in area surrounding implant can be high and fatal for the modelling/remodeling of bone which is a

ARPN Journal of Engineering and Applied Sciences

www.arpnjournals.com

key factor for the success or failure of a dental implant [6]. The effect of size of implant; diameter, indicated that the bigger implant size will increase contact surface which is reducing the stress pattern. Simultaneously, by increasing the contact surface, the stability of implant also will increase [7]. The better coupling between cortical with implant surface especially for the immediate placement will help to reduce the stresses accumulation due to loading process [8].

The effect of densities variation on stress distribution have been investigated by Premnath *et al* [9]. They generated four of models to simulate a single threaded implant placed in four different densities of bone (D1, D2, D3 and D4) where the Poisson's ratio and Young's modulus of elasticity of the material were incorporated. The generated stresses are measured in the form of maximum Von Mises stress which is measured in the different location. For same vertical force they are used, the density effect is only significant affected the stress level in the implant body and cortical bone only. The cancellous bone is not affected significantly.

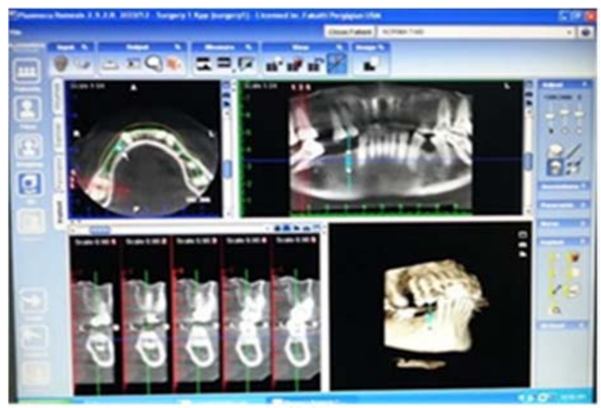
This paper discussed the biomechanical assessment on how the cortical thickness affects the stress distribution on dental implant system. The methodologies of our technique that are used in this research are explained in the following section.

METHODOLOGY OF THE STUDY

The ethical approval for this study was obtained from the Human Research and Ethics Committee of University Sains Malaysia (USM) [No.254.4(1.3)]. One of dental implant patient has been involved with the criteria: Malay race, age between 25 - 60 years old, natural edentulous case, has edentulous at molar teeth. Meanwhile the exclusion criteria are: Non - smoker and the person who has not systemic disease such as heart problem, high blood pressure and diabetic.

The patient was scanned with the Planmeca 3D Cone Beam System follows the standard procedure in HUSM CBCT lab. A radiograph screening has been conducted by clinician an indication of overall anatomy of the maxilla and mandible and potential vertical height of available bone; anatomical anomalies or pathological lesions; sites where it may be possible to place implants without grafting and sites that would require grafting; restorative and periodontal status of remaining teeth; length, shape, angulation, and proximity of adjacent tooth roots. The illustration of CBCT scanning for this evaluation purposes is shown in Figure-1.

(a)



(b)

Figure-1. (a) The position for CBCT scanning of the dental implant patient in HUSM, with the exposure: 84 kVp, 8 mA, 320-μm voxel resolution, and FOV 16 cm. (b) the recorded images are saved in the database system and it will be transferred into MIMICS software for segmentation step.

On the day of surgery, the patient is administered topical and local anaesthesia. The flapless procedure is started using the dental drill, cutting the tissue into a tissue plug which is removed. The 2.0 mm drill guide is inserted into the guide sleeve and the 2 mm twist drill with a drill stop set to depth is used to begin the osteotomy. The 3.2 mm drill guide is inserted into the guide sleeve and the 3.2 mm twist drill with a drill stop set to depth is used to enlarge the osteotomy. The gum tissue is then opened in the area of the implants to expose them. The Mega Gen 4.0 x 11.5 mm implant is mounted onto the implant driver and placed to the planned depth and torque. For this placement of implant the sub merger type where healing screw would close (suture) restorable about 3 mm was used. The illustrations of implant placement for molar edentulous are shown in Figure-2

VOL. 12, NO. 14, JULY 2017 ISSN 1819-6608
ARPN Journal of Engineering and Applied Sciences

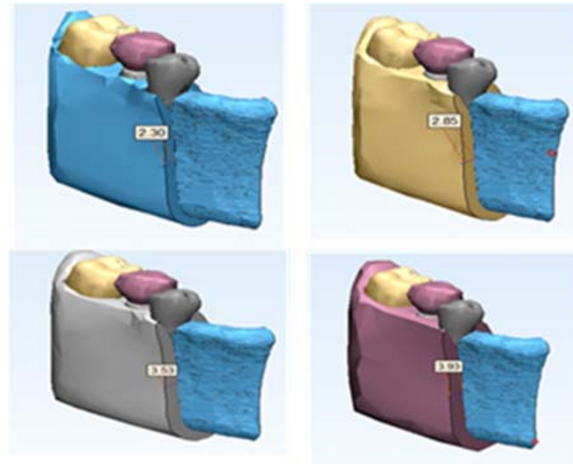
www.arpnjournals.com

(a) (b)

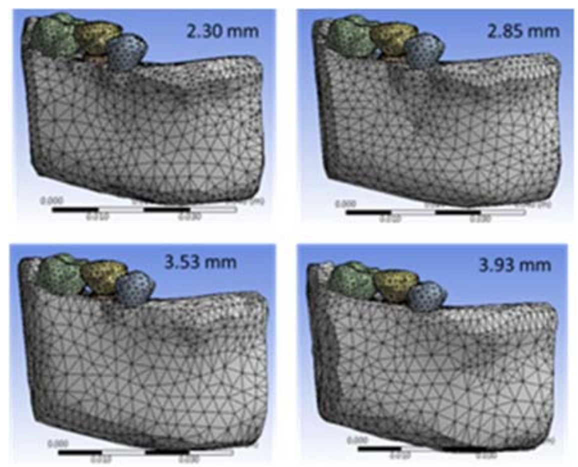
(c)

Figure-2. (a). The lost of molar toot, (b). The placements implant insertions, and (c). The torqueing wrench/screwdriver instrument being used to tighten the implant.

The MIMICS software Version 17.0, manufactured by Materialise NV Technologielaan 15 BE-3001 Leuven was used in segmentation. The segmentation steps are started by separating the hard tissue from soft tissue using threshold technique on HU of CBCT data. The cut off of HU for this separation is determined based on the contrast of the hard tissue with soft tissue. The Region of Interest (ROI) is cropped to get only an area around the implant and neighbour teeth only. On this bulk of ROI, every element that consist of two neighbour teeth, bone, implant and crown were segmented using Boolean operation technique. In the final stage, the morphology operation was performed to get the variation in the cortical thickness. The four different cortical thickness and other components obtained from segmentation are shown in Figure-3.



(a)



(b)

Figure-3. (a) Models of dental implant system with different cortical thickness of: 2.30 mm, 2.85 mm, 3.53 mm, and 3.93 mm. Each model consisted: crown, implant, two neighbour teeth, cortical and trabecular bone. (b) Meshing of each model.

After all the geometries of the dental system are obtained, all the components are discretised to create the mesh by using ANSIS Workbench software, while the material properties of each component is shown in the Table-1.

Table-1. Material properties for material assignment during FEA study

	Density (gr/cc)	**Young modulus (GPa)**	**Poisson ratio**
Bone	2.17	13.7	0.3
Teeth	2.9	50	0.33
Implant	4.51	105	0.37
Prosthesis	6.05	70	0.19

VOL. 12, NO. 14, JULY 2017 ISSN 1819-6608
ARPN Journal of Engineering and Applied Sciences

www.arpnjournals.com

RESULTS AND DISCUSSIONS

The primary and secondary implant stability is determined by some important factors such as: the quantity and quality of bone, the technique on how the implant is inserted and the environment of mouth during recovery process. Because the level of stress and its distribution affect to the osseo integration/ remodeling process of bone, the variation of the cortical thickness is investigated whether it is influence the stress distribution particularly in the trabecular part. In this experiment, different models of dental system consisted of cortical, trabecular, body implant, crown and two neighbor teeth are used. These models are constructed from 3-D image of CBCT of dental implant patient.

Four models with different cortical thickness (2.30 mm, 2.85 mm, 3.53 mm and 3.93 mm) are constructed and used in the FEA study to understand the mechanism of stress distribution during vertical loading in the jaw system. In each simulation, the model consists of cortical, trabecular, two neighbor teeth, implant body and crown. All the parameters (meshing, boundary condition, fixed support and material assignment) during FEA study is kept the same for different model except the cortical thickness.

The stress responses due to a 200 N vertical force which is pointed out into top of crown are shown in figure-4. The level of Von Mises stress in that figure is represented by color coding. In all model, the stress was propagated intensively in the area close to the implant body and less in the area around neighbor teeth.

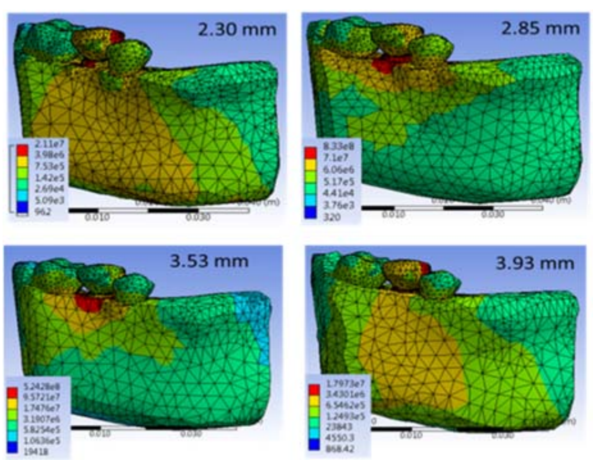
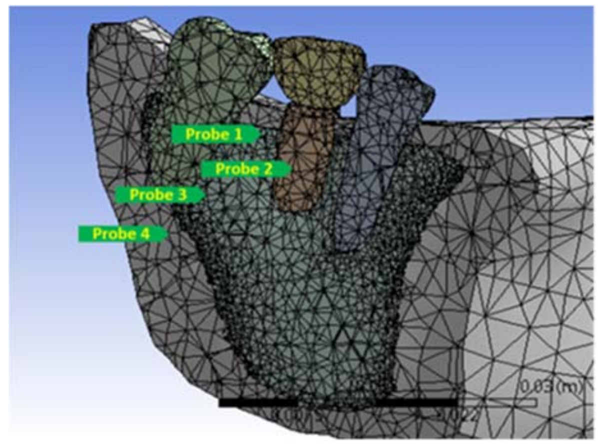


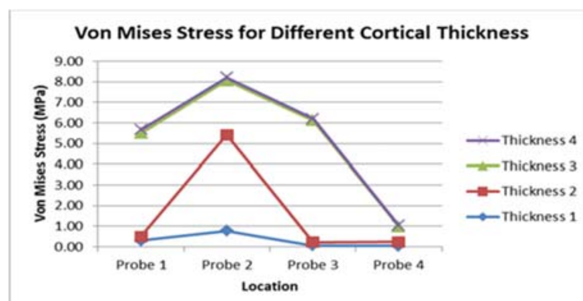
Figure-4. The Von Mises stress in 3-D view of each model with different cortical thickness.

To measure the detail of the stress responses in the jaw system, four different probe locations have been placed. Those location are: Probe 1 is in upper part of trabecular close to implant body, Probe 2 is located in the middle of implant, Probe 3 is located in the below part of trabecular bone close to the bottom of neighbor teeth, and Probe 4 is located in the bellow part of cortical. The location of the probes and measured Von Mises stress on each probes for different models are represented in the Figure-5.

The simulation results showed that if the cortical bone thickness is varied keeping the trabecular bone and teeth configuration constant, then the generated Von Mises stress with thicker cortical bone thickness become a minimum. There is a strong relationship between cortical thickness and generated Von Mises stress in the jaw system especially in the area around implant location.



(a)



(b)

Figure-5. (a) Probes location (m), and (b) Von Mises Stress (MPa) measured at each of the probe with different cortical thickness.

CONCLUSIONS

Four different cortical thickness models have been constructed from in vivo CBCT data to investigate the influences of cortical thickness on stress distribution due to the vertical force loading. FEA analysis has been used to analysis the generated Von Mises stress on each model with different cortical thickness. Based on our measurement, the stress distributed in whole area of the jaw system is affected by cortical thickness. The cortical thickness plays an importance role in the stress distribution. Model with thicker cortical, the stress will be propagated easily into trabecular, and hence the trabecular will undergo higher stress compare with thinner cortical thickness. Continuous loading as resulted by mastication process during healing stage need to be taken into account to avoid overload on the new implant inserted that will disturbed the modeling/remodeling the bone, hence the long lifetime of dental implant can be achieved.

Based on the results, it is important to consider the factor of generated stress on different cortical thickness into implant design to minimize the generated Von Mises stresses. It might be solved by selecting the proper implant material for different cortical bone thicknesses,

ACKNOWLEDGEMENT

The authors are grateful to USM and UiTM for funding this research with the grant number: PRGS: 1001/PPSG/8146004 and Short term grant: 304/PPSG/61313004; and 600-RMI/FRGS 5/3 (142/2014.

REFERENCES

[1] P. Be, A. Ag, M. Zwahlen and I. Sailer, “Improvements in implant dentistry over the last decade : comparison of survival and complication rates in older and newer publications,” Int J oral Maxillofac Implant. Vol. 29, pp. 308-24, 2014.

[2] J. P. Geng, K. B. Tan and G. R. Liu, “Application of finite element analysis in implant dentistry: a review of the literature.” J. Prosthet. Dent. Vol. 85, no. 6, pp. 585-598, 2001.

[3] H. Y. Chou, J. J. Jagodnik and S. Müftü, “Predictions of bone remodeling around dental implant systems,” J. Biomech. Vol. 41, no. 6, pp. 1365–1373, 2008.

[4] H. Zamanian, B. Marzban, M. Gudarzi and S. Amoozegar, “Investigation of the Effect of Dental Implant Screw Pitch on the Stress and Strain Distribution in the Mandibular Bone,” in Corference paper, 2012, December.

[5] G. Sivamurthy, “Stress distribution patterns at mini-implant site during retraction and intrusion - a three-dimensional finite element study,” Prog. Orthod. pp. 1-11, 2016.

[6] N. Djebbar, B. Serier, B. B. Bouiadjra, S. Benbarek, and A. Drai, “Analysis of the effect of load direction on the stress distribution in dental implant,” Mater. Des. Vol. 31, no. 4, pp. 2097-2101, 2010.

[7] R. Yeşildal, F. Karabudak, F. Bayındır, and H. Zamanlou, “Effect of Implant Diameter and Length on Stress Distribution for Titanium and Zirconia Implants by Using Finite Element Analysis (FEA),” Open Access Libr. Journal, 2 e1211. http//dx.doi.org/10.4236/oalib.1101211, 2015.

[8] B. Saluja, M. Alam, T. Ravindranath, a Mubeen, N. Adya, J. Bhardwaj and a Dhiraj, “Effect of length and diameter on stress distribution pattern of Indident dental implants by finite element analysis,” J. Dent. Implant. Vol. 2, no. 1, p. 19, 2012.

[9] K. Premnath, J. Sridevi, N. Kalavathy, P. Nagaranjani, and M. R. Sharmila, “Evaluation of stress distribution in bone of different densities using different implant designs: A three-dimensional finite element analysis,” J. Indian Prosthodont. Soc. Vol. 13, no. 4, pp. 555-559, 2013.