3D-FEA of Implant Thread Depth and Pitch on Interfacial Stresses

Yulin Zhou, Zhihong Mao*, Fan Liu and Ruichao Wang

Division of Intelligence Manufacturing, Wuyi University, China

*Corresponding author

Abstract. Objective: Using the numerical analysis, the interfacial stresses with various thread depths and pitches implants are evaluated in mandibular bone mass. Methods: Implant parameters considered herein include the thread depths of 0.30, 0.35, 0.40, 0.45, 0.50 and 0.55 mm and the pitches of 0.75, 0.80, 1.0 and 1.25 mm. A simplified bone-implant system is created by Geomagic studio and Solidworks. ANSYS workbench was used to analyze the effects of the stress distribution. Results: The maximum von Mises was exhibited in three position at the implant: the location between implant and abutment, the area between implant and screw and the threaded implant body (connect area between first thread and second thread or second thread and third thread). whereas uneven stress distribution were showed in both cortical bone and trabecular bone under oblique and vertical loading. Conclusions: This study found that the force (magnitude and direction) is the important factor of influence on stress, for biomechanical context, the ratio of the thread depth to the pitch at the implant below 0.55 is more appropriate for controlling the max EQV of implants.

Keywords: Implant; Thread depth; Pitch.

1. Introduction
Dental implants have been used in clinical dentistry, which are surgically implanted directly into the bone tissues representing missing teeth or a tooth. Thus, load transmission (from implants to surrounding biologic tissues) is important for implant restorations[1]. But from a biomechanical point of view, which are the optimizing range of stresses has so far proved inconclusive. Two main hypothese theorized are generally considered. Wolff stated that the bigger the stresses, the more bone formation there are, while the less stresses, the more bone loss there are. Another influencing view is that extreme stresses may cause bone loss[2-3]. The stresses of dental implant and peri-implant bone tissues are affected by various factors related to the configuration of implant, bone quality, treatment and so on[4-5]. Lucie Himmlova et al stated that implant diameter, length influence stress distribution, lots of von Mises equivalent stresses appeared at bone area around implant neck[6]. But dental implant models used in this research did not combine thread design, which resulting in possible bone-to-implant contact area (BIC) change. The stability of implant is affected by BIC[4]. Thread designs function to transfer the load or connect with other parts. V-thread implant improve primary implant stability with ins better initial mechanical interlocking. Rectangular thread implant might transfer vertical force evenly than V-thread, but with the similar von Mises equivalent stresses. The buttress thread implant was designed for combining the advantage of V-thread and rectangular thread implant[6]. Therefore, the configuration of thread need to be considered for facilitating dissipation of von Mises equivalent stresses at the bone-implant interface or minimum peak stress. The configuration of thread including thread shape and size (pitch and thread depth), although thread shape...
affected stresses has been investigation[7], further investigations for the influences of various pitch and thread depth at the stresses in the bone-implant interface is required.

Finite element analysis (FEA) is generally considered a used tool for simulating the loading condition of implantation, which evaluate the biomechanical properties of implant and peri-implant bone tissues in mimic the complexities of clinical situation[8]. FEA is a approximation numeric analysis tool, the accurate of results may affected by realistic fundamental assumptions and a near actual physical model. In brief, The more accurate the model is, the closer the results are. 3-D model can make a more specific and accurate description of the clinical phenomenon than 2-D model[1]. In this study, a 3-D FEA were used to explain the effect of various thread depth and pitch on interfacial stresses.

2. Materials and Methods

2.1. Generation of implant models

The original thread implant with 4.1mm diameter and 10mm length was created using Solidworks software. Bending deformation or deformation caused by shear loading have been reported at the most detrimental condition for bone tissues[4]. Therefore, The range of thread pitch under consideration in bending deformation was chosen according to Yamamoto analytical method[5].

By assuming the V-thread as a cross-section of symmetrical beam, the bending deformation $\delta$ of V-thread was obtained by Equation (1),

$$\delta (x) = \frac{F_x}{E I} = \frac{12 [(c - x) F \cos \beta - \frac{h b F \sin \beta}{2}]}{E (a - 2 x \tan \beta)^3}$$

Where, Figure 1 is the cross sectional view of V-thread:

- $F$ is the applied load;
- $J$ is the moment of inertia of V-thread section;
- $E$ is the elastic modulus of V-thread implant;
- $\beta$ is flank angle;
- $a$ is the bottom edge length of cross-sectional view of V-thread;
- $b$ is the longitudinal displacement in the loading position;
- $c$ is the transverse displacement in the loading position;

![Figure 1. The cross sectional view of V-thread.](image)

Also, the present study calculated the extreme thread depth $h_{TD}$ of V-thread by the following Equation(2):

$$h_{TD} = \frac{12 \tan \beta}{5P - 0.6}$$

The reference maximum value of pitch and thread depth were needed to design comparison values in this study. Therefore, twenty-four implants comprising four pitches (0.75, 0.80, 1.0 and 1.25 mm) and six thread depths (0.30, 0.35, 0.40, 0.45, 0.50 and 0.55 mm) were constructed by SolidWorks software. Figure 2 is the schematic with respect to dental implant parameters. Depending on pitch and thread depth of dental implants, implant models are identified by “pitch(P)-thread depth(TD)”. “P1.0-TD0.30” is the symbol indicates the dental implant with the pitch of 1.0mm as well as the thread depth of 0.30mm.

2
2.2. Numerical simulation
This study is based on three dimensional finite element analysis (3-D FEA). A detailed finite element model of a simplified bone segment with implant-abutment system and a dental crown was obtained by SolidWorks software and imported into ANSYS workbench. Frictional contact are allowed between implant and bone (cortical bone and trabecular bone) with the friction coefficient of 0.4, whereas the contact were used to simulate the initial state after placement. There was 100 N preload between implant and screw. ANSYS software was used to divide mesh automatically with a maximum element size of 0.35 mm, which ensuing mesh quality[1]. The total number of elements was obtained with a range from 228630 to 270305, while the total number of nodes was obtained with a range from 393858 to 464651. The materials were assumed to be elastic, isotropic and homogeneous, deduced from literature[9], are reported in Table 1.

| Material           | Young’s modulus $E$ (Mpa) | Poisson’s ratio $\mu$ |
|--------------------|---------------------------|-----------------------|
| Ti                 | 110,000                   | 0.35                  |
| Cortical bone      | 13,700                    | 0.3                   |
| Trabecular bone    | 1,370                     | 0.3                   |
| Porcelain          | 70,000                    | 0.19                  |

The bottom region of bone was fixed as the boundary condition when simulating bone biomechanics. For the physiologic masticatory loads, the models was statically loaded vertically with 100 N and loaded obliquely with 100 N at 30 degrees to the long axis of the dental crown. Maximum equivalent von Mises stress (Max EQV) can be set as a measure of potential damage positions. And the stress distributions along the bone-implant interface was calculated in this study.

3. Results

3.1. A comparison of the implants
The stress distribution on bone-implant interface were shown in Figure 3.
Figure 3. Typical stress distributions in this study.
The von Mises equivalent stress of implants with various pitch and thread depth showed a stress gradient (from up to down) along the load direction, where red represents the maximum von Mises equivalent stress (max EQV) and dark blue represents the minimum von Mises equivalent stress (min EQV). Also, the maximum von Mises was exhibited in three position at the implant: the location between implant and abutment, the contact area between implant and screw and the thread of implant body (connect area between first thread and second thread or second thread and third thread). Under oblique loading, the maximum value of max EQV appeared in implant thread with the pitch of 0.75mm and the thread depth of 0.55mm (P0.75-TD0.55). The detailed value was 361.84MPa. And the others ranging from 272.79MPa to 349.85MPa. The stress magnitudes of implant thread were higher than other two site, and the value was 137.12 MPa in P0.75-TD0.55 at vertical loading, while the von Mises equivalent stress values of implant ranging from 117.35 to 123.83MPa in each other.

Figure 4. Relation between the max EQV of implant and ratio $k$.
The $k$ value was the ratio of the thread depth to the pitch at the implant (Figure 4). The max EQV increment continued to increase for $k$ in oblique loading. In vertical loading, the P0.75-TD0.55 implant showed the maximum max EQV with $k = 0.733$, while others with $k$ ranging from 0.24 to 0.563 showed a substantially lower effect of Max EQV. The plotting of max EQV in $k$ for implant models indicated by Figure 3.3. In this study, some $k$ values with multiple values had been considered in a maximum max EQV.

3.2. A comparison of peri-implant bone tissues
The results demonstrated an uneven stress distribution in both cortical bone and trabecular bone. As shown in Figure 3, the area of max EQV (red color area) was not only exposed to the location around the neck of the implant, but showed in the middle irregular part of cortical bone, which scattered from center. The location of max EQV was difference for trabecular bone with various implant thread
designs. The max EQV acting in the trabecular bone around the contact area between the first and second thread of implant or the contact area between the second and third thread of implant. Further concentrated stress area inside the trabecular bone exposed to the location around the bottom of the implant or around the area underpart of implant thread.

Under oblique loading, a comparison of the max EQV in all bone models. The stress levels for cortical bone were in the range of 54.771 to 60.354MPa, while the stress levels for trabecular bone were in the range of 6.107 to 9.733MPa. The maximum and minimum max EQV occurred respectively at the two positions. Under vertical loading, the thread size effect on the stress levels and the location of max EQV in the cortical bone, cortical bone around P0.75-TD0.35 implant provided the lowest max EQV at 8.356MPa, whereas cortical bone around P0.75-TD0.4 implant provided the highest max EQV at 10.647MPa. And trabecular bone around P0.8-TD0.5 implant provided the lowest max EQV at 3.154MPa, whereas trabecular bone around P1.0-TD0.5 implant provided the highest max EQV at 4.491MPa.

4. Discussion

This study focus on the interfacial reactions. When stress distributions in implants were compared, max EQV was found in three position, which will showed three dangerous locations in term of the fourth strength theory of material. A excessive max EQV of material will cause permanent deformation. All of the max EQV in this study were well below the yield strength of titanium alloy with 870MPa[9], whereas the maximum max EQV of implant was showed in the thread of implant body (connect area between first thread and second thread or second thread and third thread) with the pitch of 0.75mm and the thread depth of 0.55mm. This indicates that the implant will easily yield under this position. One of possible reason is that with the increase of thread depth, the area of implant excluding thread becomes smaller, which is consistent with those of Shi bin et al[10]. Another possible explanation for a max EQV was shown in the area between two thread is associated with pitch, which in consistent with previous study[11]. $S_M$ is the surface area of clearance between threads, $F$ is the force applied to the implant body, and $\sigma_M$ is the stress between the thread, which may be expressed as Equation(3). $d_\theta$ is diameter of thread, $h$ is the height of implant body, while $S_M$ can be calculated as Equation(4).

$$\sigma_M = \frac{F}{S_M}$$  \hspace{2cm} (3)

$$S_M = \pi h(d_\theta - 2h_{in}) - \frac{2\pi h \tan \beta}{p}(d_\theta h_{in} - h_{in}^2) - \frac{0.1h \pi (d_\theta - 2h_{in})}{p \cos \beta}$$  \hspace{2cm} (4)

Where, $d_\theta$ is the diameter of thread, $h$ is the height of implant body (see Figure 2).

Studies have indicated occlusal force(magnitude and direction) play a major role in stresses[12]. It is known from non-linear finite element analysis that the loading type is a important factor for the stress distributions, and stresses under oblique loading showed larger stresses than under vertical loading[12]. In this study, the max EQV of implant in oblique loading was two to three times as great in vertical loading, whereas the max EQV of cortical bone in oblique loading was about ten times the value of the cortical bone in vertical loading and the max EQV of trabecualar bone in oblique loading was about two times as great in vertical loading. These results are consistent with that of previous studies[8-9].

When the force has been fixed, the surface area of clearance between threads $S_M$ need to be considered for obtaining a more appropriate implant thread stresses $\sigma_M$. The Equation (4) showed that increasing in pitch and decreasing in thread depth may significantly decrease the surface area of clearance between threads, which will increase the $\sigma_M$ with the same $F$ in the implant body. However, this study found that among the implants, the ratio $k$ was also associated with the $S_M$.The results have shown that the max EQV increase for increasing $k$ value, the value excess 0.55 can shown a larger max EQV. This likely reason from this result that max EQV inside the implant is not sure, therefore, the larger max EQV in implant is located the contact area between thread of implant body. The max EQV exhibited in thread (only appeared in P0.75-TD0.55 implant in this study) was a less frequent situation but
dangerous complication than other mechanical complications[13]. Based on this, the extreme value of $k$ is known from $\frac{ds_m}{dh_m} = 0$, then $k$ can be calculated as follow:

$$k = \frac{d}{2P} + \frac{1}{2\tan\beta} - \frac{0.1}{2P \sin\beta}$$

(5)

The ratio $k < 0.55$ is a relatively optimal selection for thread designs by controlling the max EQV. Not all of the thread implant design parameter $k$ exhibited lower max EQV as compared to $k > 0.55$ thread implant. For other thread parameter applications, each thread has its extreme value $k$ with regard to the diameter of thread $d_0$, flank angle $\beta$ and pitch $P$.

5. Conclusion

This study found that the force (magnitude and direction) play a significant role in stress, for biomechanical context, the ratio of the thread depth to the pitch at the implant below 0.55 is more appropriate for controlling the max EQV of implants. In additional to the value of max EQV, the position of the max EQV was also important. The max EQV appeared in clearance between implant thread was more dangerous than other locations.

Acknowledgment

This study was supported by the Innovation projects of Department of Education of Guangdong Province, China (NO.2017KTSCX183) and the Funding for Introduced Innovative R&D Team Program of Jiangmen(Grant No.2018630100090019844).

References

[1] Alireza M, Alireza Z, and Mahmoud K., “The Effect of Implant Thread Design on Stress Distribution in Anisotropic Bone with Different Osseointegration Conditions: A Finite Element Analysis,” The International Journal of Oral & Maxillofacial Implants, vol. 30, pp. 1317-1326, Nov 2015.

[2] Abuhussein H, Pagni G, Rebaudi A, and Wang H L, “The effect of thread pattern upon implant osseointegration: Review,” Clin Oral Implants Res, vol. 21, pp. 129-136, February 2010.

[3] Hsu J T, Shen Y W, Kuo C W, and Wang R T, “Impacts of 3D bone-to- implant contact and implant diameter on primary stability of dental implant,” Journal of the Formosan Medical Association, vol. 116, pp. 1-9, August 2017.

[4] Misch C E, “Dental Implant Prosthetics,” Journal of Prosthodontics, vol. 14, pp. 212-214, 2005.

[5] Yamamoto A, “The theory and computation of threads connection,” In: Keqian Guo, Sujuan Gao, Xiaofeng Wang, Aling Xu translated. Shanghai Scientific and Technological Literature Press, Shanghai, pp. 45-61, 1984.

[6] Himmllová L, Dostálková T, Kácovský A, and Konvicková S, “Influence of implant length and diameter on stress distribution: A finite element analysis,” Journal of Prosthetic Dentistry, vol. 91, pp. 20-25, January 2004.

[7] Alireza M, Alireza Z, and Mahmoud K, “The Effect of Implant Thread Design on Stress Distribution in Anisotropic Bone with Different Osseointegration Conditions: A Finite Element Analysis,” The International Journal of Oral & Maxillofacial Implants, vol. 30, pp. 1317-1326, October 2015.

[8] Mao Z, Yi D, and Cao G, “Influence of sizes of abutments and fixation screws on dental implant system: a non-linear finite element analysis,” Biomedizinische Technik/Biomedical Engineering, vol. 62, pp. 357-364, 2017.

[9] Bouazza-Juantes K, Martinez-Gonzalez A, Peiro G, Ródenas J, and López-Mollá M, “Effect of platform switching on the peri-implant bone: A finite element study,” Journal of Clinical and Experimental Dentistry, vol.7, pp. 483-488, November 2015.

[10] Shi Bin, and WU Tao, “Technical complications associated with implant prostheses in terms of reason, prevention, and management,” Journal of Prevention and Treatment for Stomatological Diseases, vol. 26, pp. 415-421, 2018.
[11] Hansson S, and Werke M, “The implant thread as a retention element in cortical bone: the effect of thread size and thread profile: a finite element study” Journal of Biomechanics, vol. 36, pp. 1247-1258, October 2003.

[12] Lan T H, Du J K, Pan C Y, Lee H E, and Chung W H, “Biomechanical analysis of alveolar bone stress around implants with different thread designs and pitches in the mandibular molar area,” Clinical Oral Investigations, vol.16, pp. 363-369, February 2012.

[13] Geng J P, Ma Q S, Xu W, Tan K B, and Liu G R, “Finite element analysis of four thread-form configurations in a stepped screw implant,” Journal of Oral Rehabilitation, vol. 31, pp. 233-239, Finite element analysis of four thread-form configurations in a stepped screw implant 2004.