Effects of Dental Implant-abutment Interfaces on the Reliability of Implant Systems

Xiao Zhang¹, Lei Liu^{1,2}, Yaling Wang¹, Xianshuai Chen^{1,3}

¹ Shenzhen Institutes of Advanced Technology, Shenzhen, China, 518055

² College of Mechanical Engineering, Shanghai University of Engineering Science, Shanghai, China, 201620

³ Foshan Angels Biotechnology Co., Ltd., Foshan, China, 528000

Abstract. In this paper, by analyzing the effects of two different kinds of implant-abutment connection interfaces under the same working condition on the mechanical and fatigue performances of the implant system as well as on the surrounding bones, we intend to study such effects on the reliability of the implants and provide a theoretical basis for the design and clinical application of dental implant systems. For the purpose, we adopt a 3-D modeling method to establish the model, and use FEA (finite element analysis) to carry out static mechanic and fatigue analysis on the implant system and its surrounding bones; then we make the two implant systems, and carry out fatigue tests on a dynamic fatigue testing machine to verify the FEA results. After comparing the results from the two different systems, we find that the stress distribution and fatigue safety factor of the system which has deeper axial matching of the taper connection are better than those of the other system, that is to say, between the two major elements of a implant system, the axial length of the connecting taper and the size of the hexagon, the former has greater effects than the latter. When the axial matching is deeper, the stress distribution of the implant system will be better, the fatigue safety factor will be higher, and the implant system will be more reliable.

1 Introduction

Many factors may influence the success of dental implantation; however, the reliability of the implant system itself is one of the most important [1-4]. At present, a dental implant system is generally assembled of the implant, the abutment and the central screw, forming a two-part structure. The reliability of an implanted system lies mainly in two factors: the force bearing capacity and the fatigue resistance during continuous bites. Therefore, a good design of implant system is bound to make the system get a better stress distribution and a greater fatigue safety factor under forces. The implant-abutment connection interface, i.e. the connecting and contact means between the implant and the abutment, is therefore the key factor of the reliability of the implant system. Today most of the relatively good products on the market adopt the implant-abutment connection interface that uses a taper surface and a hexagon connection matching [5, 6]. However, there is no clear conclusion regarding how to optimize the interface design and what kind of factors having the greater effect on the reliability of implant systems.

On the basis of the research on current designs, in this paper, we carry out analysis on the effects of two different kinds of implant-abutment connection interfaces on the stability of the systems, so as to optimize the design and make choices between the two systems. First we use the CAD software SolidWorks to establish two

implant system models and the corresponding alveolar bone models based on two different kinds of implant-abutment connection interfaces, then we apply ANSYS Workbench to conduct FEA analysis on the systems and alveolar bones for static force analysis and cycle fatigue analysis, and finally we make sample pieces and carry out dynamic fatigue test to study the stress distribution and fatigue resistant characteristics of the systems, and verify the influencing factors of the implant-abutment connection interface on the reliability of implant systems.

2 Design of the Dental Implant-Abutment Connection Interface and the Implant System

In the design of dental implants, we mainly take into account the interior hexagon angle positioning, the platform transfer and the Morse tapered connection. The connection interface between the implant and the abutment is just the connection between the hexagon and the taper. Figure 1(a) shows the connection interfaces of #1 implant system and #2 implant system as well as their sizes. The taper angles of both systems are 24°, the axial match heights are respectively 1.25mm and 1.65mm, the distances between opposite sides of the hexagon are respectively 2.64mm and 2.5mm, and the length and diameter of both implants as well as other dimensions of the abutments are the same. That is to say, the two

implant systems mainly differ in the axial height of the taper and the size of the hexagon.

The exterior thread of the implant is designed to be trapezoidal, with the pitch (P) of 0.6mm, the diameter of 4.3mm and the length of 11.5mm, while the diameter of the abutment neck is 5mm; the central screw is 8mm long, with standard thread M2. The size of this design is the most commonly used on the market [6, 7]. The crown bears the main load. The simulated crown consists of a cylinder and a hemisphere: the cylinder is 10mm in diameter and 5mm in length, while the hemisphere is 10mm in diameter [8]. The modeling of the shape of the alveolar bone tissue is based on the CT scanning data of human lower jaws, including two parts: the cortical bone and the cancellous bone. The crown is wrapped with the cortical bone, which is 2mm thick [6]. By the SolidWorks software, we now establish the component models and analytical models above, as shown in Figure 1(b)(c).

Figure 1. (a) Implant-abutment connection interface; (b) (c) Diagrams of implant system analysis models

3 Finite element analysis (FEA)

3.1 Materials and Methods.

Through the CAD interface of the ANSYS software, we import the file into the FEA software ANSYS Workbench for static and fatigue analysis. In this research we assume that all the tissue and materials in the models are continuous, homogeneous and isotropic linear elastic materials. The materials for the implant systems are titanium alloy Ti6Al4V for its best biocompatibility, and the crowns adopt zirconia [6, 9]. Specific material properties are shown in Table 1.

Table 1. Material Properties

Material	The young's modulus (E, GPa)	Poisson's ratio (V)
Cortical bone	14	0.30[6]
Cancellous bone	3	0.30[6]
Ti6Al4V	110	0.35[6,9]
Zirconia	70	0.28[6,9]

3.2 Static Analysis and Fatigue Analysis on the Implant Systems

This FEA simulates the working conditions under which the implant system and the alveolar bone are in full osseointegration. The lower of the alveolar bone is a fixed constraint, the contact between the implant and the abutment, and the contact between the alveolar bone and the implant are both assumed to be fixed [5, 6]. The force direction is an angle of 30 degrees from the axial direction of the implant, and in the test we use a force of 200N, with a pre-tightening torque of 30N.CM on the central screw [5-7]. What the working condition for fatigue analysis is that under the conditions defined by the above constraint and contact, in the force direction, load the forces from 20N to 200N in sine waves, with the amplitude of 90N and frequency of 15Hz. An acceptable sample should not be damaged or yield under 5×10^6 times of cyclic loading [8], as shown in Figure 2.

Figure 2. The FEA model and the working condition of an implant system and the alveolar bone

3.3 Results

An implant system, which has better stress distribution and greater fatigue safety factor under loading, will have better stability. As the implant-abutment connection interface is the relatively weak part of a system, so we focus on the stress distribution and the fatigue safety

factor of the interface, and through the comparison analysis on the two interfaces, we study their effects on the stability of the whole implant system as well as on the stresses of surrounding bones. The two kinds of interfaces and their equivalent stress contours are shown in Figure 3, while the results of analysis on fatigue safety factors are shown in Figure 4.

As can be seen from Figure 3, the maximum stress of #1 implant system is 289MPa, occurring at the edge of the taper of the implant-abutment connection interface. The minimum stress at the implant-abutment connection interface is 1.8MPa, where which at the connection between the abutment and the crown; the maximum stress of the alveolar bone is 48.3 MPa, where which at the edge of its upper end connecting to the implant. The maximum stress of #2 implant system also occurs at the edge of the taper of the implant-abutment connection interface. The maximum stress at the implant-abutment connection interface is 267MPa, and the minimum stress is 1.44MPa, where which at the connection between the abutment and the crown; the maximum stress of the alveolar bone is 48MPa, where which at the edge of its upper end connecting to the implant.

Figure 3. Two kinds of implant-abutment connection interfaces and their equivalent stress contours

Figure 4. Fatigue safety factors

As can be seen from Figure 4, the minimum safety factor of #1 implant system is 1.1, where which at the edge of the taper of the implant-abutment connection interface, and the maximum safety factor is 15; the

minimum safety factor of #2 implant system is 1.4, where which at a position the same as that of the implant-abutment connection interface of #1 implant system, and the maximum safety factor of the implant-abutment connection interface is 15.

4 Fatigue test

From the FEA analysis above, we get the diagram of effective stress distribution and the fatigue safety factor. Now we use the Willemin-Macodel 408MT 7-axis turning-milling machine center to make the two kinds of implant systems, as shown in Figure 5. Then we adopt the fatigue test method [8] which is commonly used internationally for dental implant system to carry out the fatigue test. Figure 6 shows the fatigue test program and the test site. The test machine is BOSE's dynamic fatigue test machine, 3510-AT, and the force sensor precision is $\pm 0.5\%$. The rigid fixture is used to position and clamp the implant. The head bearing device is a stainless steel hemispherical cap, with a size the same as the crown in the FEA analysis. There is no constraint in the horizontal direction at the contact position between the loaded parts and the hemispherical cap, with the loaded force, frequency and cycle period the same as in the FEA working condition.

Figure 5. Components of an implant system

1. Loading device; 2. Nominal bone level; 3. Connecting part; 4. Hemispherical loading member; 5. Dental implant; 6. Specimen holder; 7. $L=1\text{mm}$; $y=1\sin 30=0.5\text{mm}$.
Figure 6. Fatigue test program and the test site

The test results are shown in Table 2 below. #1 implant system is damaged, as shown in Figure 6, while #2 results in no damage. As can be seen from the comparison of FEA analysis results, #1 implant has a lower safety factor, although not yet reaching a degree of damage. The damage of the implant-abutment connection interface in #1 implant system during the test may relate to the taper matching precision, yet the overall results are consistent with the FEA analysis results, that is to say, the damaged part is basically the same as the position of the maximum stress at the implant-abutment connection interface in the FEA analysis. This not only further demonstrates the feasibility of the FEA method, but also proves that the #2 implant-abutment connection is better.

Table 2. Test experiment results

	Load (N)	Cycle(times)
#1 implant system	200	3,934,006
#2 implant system	200	5,000,000

5 Discussion

For further study of the effects of implant-abutment connection interfaces on the reliability of implant systems, we get the statistics of all the maximum equivalent stresses for the components under the above working conditions, as shown in Table 3.

Table 3. The statistics of the maximum stresses for implant systems and alveolar bone

Number	Force (N)	Force angle (degree)	Implant (Mpa)	Abutment (Mpa)	Central screw (Mpa)	Cortical bone (Mpa)	Cancellous bone (Mpa)
#1 implant system	200	30	289	221	154	48.3	9.8
#2 implant system	200	30	269	215	149	48	9.8

From the above table and figure we can learn, in the implant-abutment connection interfaces, the stress of the taper bears is larger, and the stress of the hexagon bears is comparatively smaller, which means it is the taper that bears much of the loads from the outside. Meanwhile, when comparing the features, the force distribution and the fatigue characteristics between the two implant-abutment connection interfaces, we find that the axial height of the taper matching of #2 interface is 0.4mm greater than #1 and its hexagon size is 0.14mm smaller than #1, but the implant system bears smaller stress. This indicates that the anti-pressure ability of the taper completely counteracts and exceeds the effects caused by the size reduction of the hexagon, that is to say, the taper plays a more decisive role on the reliability of the implants, and furthermore, it has more obvious effect on reducing the long-term stress of the surrounding alveolar bone.

As for the above two implant-abutment connection interfaces, #2 has a better design, its implant system has better stress distribution, fatigue resistance, reliability, and better biomechanical reaction with the surrounding bones.

6 Conclusion

To sum up, by comparing the two implant systems of different implant-abutment connection interfaces, we can draw the following conclusions:

Under the same simulated occlusal loading conditions, between the two major elements of implant-abutment connection interfaces, the axial matching depth of the taper connection has greater effects on the reliability of the implants system than the size of the hexagon. To be specific, when the axial matching is deeper, the stress distribution of the implant system will be better, the

fatigue safety factor higher, and the implant system more reliable;

The results of FEA fatigue analysis are consistent with that of simulation experiments, and can therefore provide a reference for the optimized design of implant systems.

Acknowledgments

This work is supported by the Science and Technology Program of Shenzhen, China (No. JCYJ20140617143643469), and Foshan Innovative and Entrepreneurial Research Team Program, China (No. 2014IT100102).

References

1. M. I. El-Anwar, S. A. Yousief, T.A. Soliman, et al: A finite element study on stress distribution of two different attachment designs under implant supported overdenture. The Saudi Dental Journal .Vol. 27 (2015), p.201–207
2. M. Degidi, G. Daprile, and A. Piattelli, Determination of primary stability: a comparison of the surgeon's perception and objective measurements, The International Journal of Oral & Maxillofacial Implants, vol. 25 (2010), p.558–561
3. S.-J. Ahn, R. Leesungbok, S.-W. Lee, Y.-K. Heo, and K. L.Kang, Differences in implant stability associated with various methods of preparation of the implant bed: an in vitro study, Journal of Prosthetic Dentistry, vol. 107 (2012), p.366–372
4. N. Ferrigno, M. Laureti, and S. Fanali, Dental implants placement in conjunction with osteotome sinus floor elevation: a 12-year life-table analysis from a prospective study on 588 ITI implants,

-
- Clinical Oral Implants Research, vol. 17 (2006), p.194–205
5. G. Mehdi, A. Belarbi, B. Mansouri, et al: Numerical study of effect of elastomeric stress absorbers on stress reduction in bone-dental implant interface. J Appl Oral Sci. Vol. 23 (2015), p.87-93
 6. X. Zhang,Z. Xie, et al: Effects of custom-made and thread dental implant system on the stress distribution in alveolar bone: a 3-dimensional finite element analysis. Applied Mechanics and Materials. Vols. 475-476 (2014), p.1487-1493
 7. Straumann Product Catalog, Information on: <http://www.straumann.com/>
 8. International Organization for Standardization: ISO 14801-2007 Dentistry — Implants — Dynamic fatigue test for endosseous dental implants, IHS under license with ISO (2007).
 9. L-Y.Jia: Investigation of Fatigue Properties and Fatigue Design Diagram of Titanium alloy Ti-6Al-4V. East China University of Science and Technology (2014).