Effect of Abutment Design Parameter on Mechanical Properties for Dental Implant System

Xiuyuan Lu$^{1,2}$, Xiao Zhang$^{1,2,3}$, Jianyu Chen$^2$ and Xianshuai Chen$^2$

$^1$ Guangzhou Janus Biotechnology Co., Ltd., Guangzhou, 511458, PR. China
$^2$ Foshan Angels Biotechnology Co., Ltd., Foshan, 528000, PR. China
$^3$ Author to whom any correspondence should be addressed.
xiao.zhang@gzjanus.com

Abstract. Abutments, as the intermediate connection between implant and crown vary on external characters decided by design parameters in order to suit for multiple clinical cases with the development of prosthodontics. The various angle, groove height (GH) and diameter of abutment brought different magnitude of stress to the dental implant system. Excessive stress concentration lead to fatigue fracture failure. Thus, in this study, parametric design and Finite Element Analysis (FEA) were combined to understand the effects of angled-abutment GH and diameter on the mechanical properties of the implant system. 30 models of abutment with varying angles (15° and 25°), diameters (4, 5 and 6mm) and GH (1, 2, 3, 4 and 5 mm) were simulated and the values of Mises the equivalent stresses were analyzed by two-way ANOVA. The results demonstrated that the three design parameters had no influence on the area where stress concentration occurred and the diameter had no effect on the maximum stress while the GH had a significant influence ($P<0.05$).

1. Introduction
The dental implant-abutment system has been gradually approved and widely used to improve the quality of patients’ life over the past 40 years since the osseointegration was discovered by Prof. Brånemark of Sweden [1]. The planting position varies from case to case in contemporary implant dentistry, traditional straight abutments do not perfectly solve all clinical cases, which requires more specifications and types of abutments to connect implants and crowns. At present, abutment was shaped by three main parameters: angle, diameter and GH. Long-terms of stability and survival rate of implant system rely on its outstanding mechanics performance. In the process of swallowing, chewing and speaking, the implant system in mouth experienced different levels of load. Moreover, different geometric shapes of the abutments produce various levels of stress concentration [2], which may lead to cracks, and accelerate crack propagation [3]. Kao HC et al [4] demonstrated that the 25°-angled abutment increased the stress level of the surrounding cortical bone and displacement of the implant by using FEA. Clelland’s study [5-7] showed that the maximum stress of the abutment with angle of 15° and 20° were 170% and 190% compared with the straight abutment; Kebin Tian et al [8] evaluated the influence of the titanium alloy angled abutments’ geometry on the stress distribution by means of FEA and the results showed that angled abutments resulted in decreased stresses when implants were not placed in ideal axial position. However, little research has been focused on how the GH and diameter affects the stress level of the implant system. Therefore, a virtual prototype needs to be built and simulated to understand the influence of different design parameters on the mechanical properties of the whole implant system.
This paper focuses on the effects of design parameters of the abutment on the stress level and distribution of the implant system.

2. Material and methods

2.1. 3D models
The dental implant-abutment system consisted of implant body, central screw and abutment. It was proved that the smallest diameter of implant (D=3.5mm) and central screw (M1.8) caused the maximum stress [9]. In the meanwhile, implants with same diameter but different length (L) showed little effect on the stress level [10]. Therefore, the implants (D=3.5mm, L=10mm) and the central screws (M1.8) were used in this study in order to amplify the stress concentration under extreme conditions.

The implant body and central screw was simplified to cylinder in the case that all the thread connections were ideal and valid to focus on the regular of stress for some study have suggested that cylindrical implant models might be a good approximation of screw-type implants in further FEA [11].

2.2. Experimental design
The abutment GH was the sum of the depth of the gum combined with the length of the epithelium and attachment of the connective tissue, additionally groove surface needed to fit the surgical wound of gum. Therefore, the ideal GH was 2.0-5.0 mm [12] and ideal diameter was 4-6mm, and generally abutment angle ranged from 0-30° on the basis of surgical plan. In order to get detailed data, 30 models were divided into to two parallel groups established at 15° and 25°. The GH specifications of each group were 1mm, 2mm, 3mm, 4mm, and 5mm; the diameter specifications were 4mm, 5mm, and 6mm.

2.3. Parameterization process
By using the parameterized module of Ansys 16.0 Workbench, the solid model modified by Solidworks and geometry used in finite element analysis were connected. Prefix “DS_” needs to be added ahead the name of the dimensions which separately decide the diameter, GH and angle of the abutment. This process allows all models to be calculated all at once without resetting boundary conditions and contact condition. Solid model and parameters range are shown in Figure 1.

2.4. Finite element analysis (FEA)
A standardized implant fatigue testing method (ISO 14801) [13] declares a detailed Implant system test model as shown in Figure 2. Material properties were shown in the Table 1. All materials were considered to be isotropic, homogeneous, and linearly elastic [14]. In this study, assembly was meshed with tetrahedral elements, which composed of 361010-495010 element and 240756-338899 nodes.
**Table 1. Material properties for FEA [9]**

| Material         | Young’s modulus (Pa) | Poisson ratio | Tensile Yield Strength (Pa) | Compressive Yield Strength (Pa) | Assignments          |
|------------------|----------------------|---------------|----------------------------|---------------------------------|-----------------------|
| TC4              | 9.6E+10              | 0.36          | 9.3E+8                     | 9.3E+8                          | Crown, abutment,      |
|                  |                      |               |                            |                                 | central screw         |
| GR4              | 9.6E+10              | 0.36          | 4.9E+8                     | 4.9E+8                          | Implant               |
| Acrylic resin    | 3.78E+9              | 0.36          | 54.6                       |                                 |                       |

Completely fixation was applied to thread contact surfaces [15], and liner connection was applied to the remaining interfaces. Additionally, a force of 250N (average bite force of adult people) [16, 17] with vertically downward direction was applied to the crown.

The final results of the finite element analysis were evaluated by the maximum Mises equivalent stresses ($\sigma_{VM}$). Stress concentration in the stress cloud map revealed the most vulnerable area in the model [18]. Two-way analysis of variance (Two-Way ANOVA) was performed on two parameters - the diameter and the GH of abutment. The significance level was set at 5%. This procedure allowed the authors to calculate the contribution of each parameter and find out the main effects.

### 3. Results & Discussion

The stress cloud diagrams revealed that stress distribution was independent of design parameters of abutment. The peak value of $\sigma_{VM}$ all appeared on abutment. Implant system with the 25°-angled abutment (GH=5mm, D=6mm) was taken as an example in Figure 3.

![Stress Distribution Diagrams](image)

**Figure 3.** The stress distribution diagrams of abutment, implant body and central screw.

The $\sigma_{VM} = 690.3$ MPa was distributed at the rounded corners of the hexagonal joint where the abutment shape change was most severe. Therefore, it could be speculated as the dangerous section. The stress distribution of implant illustrated the neck region where implant body and the upper surface of acrylic resign material contacted as well as the inner surface contacted with the region below groove surface had a higher stress concentration. The $\sigma_{VM} = 383.71$MPa was much lower than abutment.

For central screw, the stress mainly concentrated at the thread surface fitted with the first two inner thread of the implant. Since the two regions appeared same level stress concentration on the same cross section, fatigue fracture was quite likely to occur under cyclic stress.
As shown in Figure 4, for the 25°-angled abutment, the peak value of $\sigma_{VM}$ was 690.3 MPa (GH=5mm) while the nadir was 458.4 MPa (GH=1mm). And for the 15°-angled abutment, the peak was 555.4 MPa (GH=5mm) while the nadir was 414 MPa (GH=1mm).

Apparently, the $\sigma_{VM}$ of implant body, central screw and abutment increased with GH increasing when abutment has certain angle (15° or 25°). The contour lines in the 3D surface figure were almost parallel to the diameter axis, which means the diameter of abutment was foreign to $\sigma_{VM}$, and it was confirmed by two-way ANOVA. For the implant system with 15°-angled abutment, the GH had a significant influence on the equivalent stress of the abutment ($F=26.82; P=0.035<0.05$) while the diameter had no influence ($F=0.29, P=0.65$). The same result appeared when the angle was 25° that the GH has a significant influence on the equivalent stress of the abutment ($F=36.37; P=0.026<0.05$) while diameter had no impact ($F=0.068, P=0.82$).

In the case where the threaded connection (central screw-implant, implant-resin) was always valid, abutments’ GH had various extent of impact on the three primary components. Implant and abutment were less affected than the central screw. The $\sigma_{VM}$ of Central screws in 25° group was 130% compared with the counterparts in 15° group while the implant and abutment were 125%.

Theoretically, abutment with short GH and tiny angle brings optimistic effect of oral rehabilitation based on the above analysis. However, GH was decided by actual situation of clinical cases. This study suggests that if the abutment GH has to maintain the worse condition, angle and/or diameter of implant and central screw need to be reconsidered in surgical plan to decrease the stress level.

### 4. Conclusion

The aim of this study was to understand how abutments’ three parameters affect the stress distribution among the 30 abutment models. The following conclusions were drawn: stress distribution was independent of the three design parameters of abutment; The maximum stress appeared at the abutment with longest GH and bigger angle; Besides, abutment GH had significant influence on stress level while diameter had no influence; Abutment angle aggravated the influence of GH by comparison. In the further study, systematically contradistinction of GH and angle’s impact will be made to draw a rigorous conclusion.

**Figure 4.** The 3D surface diagram of each component’s $\sigma_{VM}$: (a) Abutment (25°); (b) Central screw (25°); (c) Implant (25°); (d) Abutment (15°); (e) Central screw (15°); (f) Implant (15°).
Acknowledgement

This work was supported by 1) the Science and Technology Program of Nansha District, Guangzhou (Technology Development Program, No. 2017KF007); 2) the Science and Technology Program of Guangzhou (No. 201906010032);

References

[1] Branemark P I, Adell R and Breine U 1969 Scand J Plast Reconstr Surg. 3 81.
[2] Hasan B, Roger F, Heinemann L and Keilig C 2012 Medical Engineering & Physics 34 817.
[3] Cornell K, L E E, Matthias K, J and Robert Kelly 2009 Entail Materials. 25 1419.
[4] Kao H C, Gung Y W, Chung T F and Hsu M L 2008 Int J Oral Maxillofac Implants. 23 623.
[5] Clelland N L, Lee J K and Bimbenet O C 1995 J prosthidont. 4 95.
[6] Clelland N L and Gilat A 1992 J Prosthodont. 1 24.
[7] Clelland N L, Gilat A. McGlumphy EA and Brantley WA 1993 Int J Oral Maxillofac implants. 8 541.
[8] Kebin T, Jiang C, Lilin H, Jin Y, Weni H and Dong W 2012 Medical Engineering & physics. 34 1526.
[9] Jinyang Z, Maisheng L, Xiao Z and Xianshuai C 2019 International Journal of Materials. Mechanics and Manufacturing. 7 105.
[10] Lucie H, Tat’jana D and Alois K 2004 The journal of prosthetic dentistry. 91 20.
[11] Okumura N, Stegaroiu R and E. Kitamura 2010. Journal of Prothodontic Research. 54 133.
[12] Li-na W and Zhen F 2017 Journal of oral and maxillofacial surgery. 27 290.
[13] ISO 14801:2016 Dentistry — Implants — Dynamic fatigue test for endosseous dental implants.
[14] Tao L, Kaijin H, Libo C, Yin D and Yuxiang D 2011 Applied Mathematical Modelling. 35 446.
[15] Amer M M, Rashad H A and Abdallah S 2015 Tanta Dental Journal. 12 30.
[16] Sushant M and Raosaheb P 2016 International Research Journal of Engineering and Technology. 2 850.
[17] Lang L A, Kang B and Wang R F 2003 Prosthet Dent. 90 539.
[18] Eraslan O, Sevimay M, Usumez A and Eskitascioglu G 2005 J Oral Rehab. 32 273.