한국기계가공학회지, 제17권, 제6호, pp.24～30(2018.12) ISSN 1598-6721(Print)
Journal of the Korean Society of Manufacturing Process Engineers, Vol. 17, No. 6, pp.24~30(2018.12) ISSN 2288-0771(Online)

https://doi.org/10.14775/ksmpe.2018.17.6.024

Optimization of the Groove Depth of a Sealing-type Abutment for Implant Using a Genetic Algorithm

Hyeon-Yeol Lee*, Dae-Sun Hong*,#
*School of Mechanical Engineering, Changwon National University.

유전자알고리즘을 이용한 임플란트용 실링어버트먼트의 홈 깊이 최적화에 관한 연구

이현열*, 홍대선*,#
*창원대학교 기계공학부

(Received 22 July 2018; received in revised form 31 July 2018; accepted 13 August 2018)

ABSTRACT

Dental implants are currently widely used as artificial teeth due to their good chewing performance and long life cycle. A dental implant consists of an abutment as the upper part and a fixture as the lower part. When chewing forces are repeatedly applied to a dental implant, gap at the interface surface between the abutment and the fixture is often occurred, and results in some deteriorations such as loosening of fastening screw, dental retraction and fixture fracture. To cope with such problems, a sealing-type abutment having a number of grooves along the conical-surface circumference was previously developed, and shows better sealing performance than the conventional one. This study carries out optimization of the groove shape by genetic algorithm(GA) as well as structural analysis in consideration of external chewing force and pretension between the abutment and the fixture. The overall optimization system consists of two subsystems; the one is the genetic algorithm with MATLAB, and the other is the structural analysis with ANSYS. Two subsystems transmit and receive the relevant data with each other throughout the optimization processes. The optimization result is then compared with that of the conventional one with respect to the contact pressure and the maximum stress. The result shows that the optimized model gives better sealing performance than the conventional sealing abutment.

Key Words : Optimization(최적화), Sealing-Type Abutment(실링 어버트먼트), Dental Implant(치아 임플란트), Genetic Algorithm(유전자알고리즘), Structure Analysis(구조해석)

1. Introduction

Since the concept of osseointegration, which involves the direct contact between living bone and metal of an implant, was first reported by Branemark et al.,[1] dental implant technology has advanced rapidly on the back of experimental testing and clinical applications, and dental implants are now widely used as a reliable dental treatment.

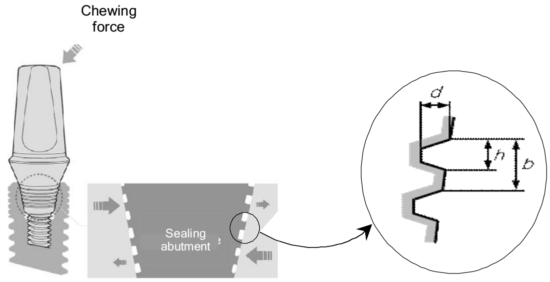
Corresponding Author : dshong@changwon.ac.kr
Tel: +82-55-213-3627, Fax: +82-55-263-5221

Fig. 1 Constitution of a dental implant

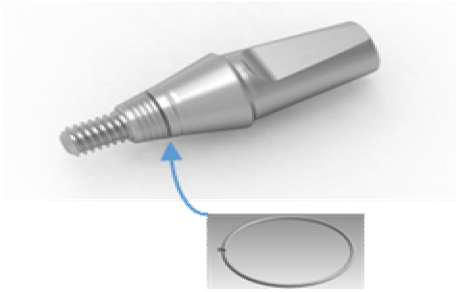
Fig. 1 shows an example of a dental implant consisting of a fixture, an abutment, and a prosthetic crown. The fixture acts a dental root implanted in the bone. At one end of the abutment is a column screwed into the fixture, which plays a role as an implant support[2]. The other end of the abutment is attached to the crown.

After implantation in the oral cavity, a dental implant replaces the role of a tooth. As such, it is subject to a continuous dynamic load and various problems can occur because of this. In particular, foreign matter can come between the prosthetic materials due to loosening between the abutment and the fixture, which can significantly degrade the functions of the implant[3,4].

A significant number of studies have been conducted to solve this problem, proposing modifications to the implant components, abutment, screw thread size and shape[5,6], chewing force, and pretension[7-9]. In particular, a sealing-type abutment has been developed to improve the contact performance between it and the fixture, characterized by the presence of a number of grooves around the circumference of the abutment (Fig. 2a)[10]. More recently, a complete sealing-type abutment[11] in which a gold ring is inserted into the top groove has been also developed (Fig. 2b).



(a) Sealing-type abutment and its groove shape



(b) Sealing-type abutment inserted with a gold ring

Fig. 2 Two types of sealing abutments recently developed

This study aimed to optimize the abutment groove depth (d) to further improve the sealing performance between the sealing abutment and the fixture, as shown in Fig. 2(a).

The performance of the new design was compared with that of an existing product using structural analysis. A series of optimization simulations was conducted using the proposed design. The simulation results demonstrated that the optimized model reduced stress when an external force was applied compared to the existing product and thus improved sealing performance.

2. Structural analysis of the abutment

2.1 Structural analysis conditions

Structural analysis was conducted on an existing sealing abutment prior to optimizing the groove dimensions. As shown in Fig. 2(a), the target product contained five grooves with a groove depth (d) of 0.05 mm, a groove height (h) of 0.1 mm, and a gap between the grooves (b) of 0.3 mm.

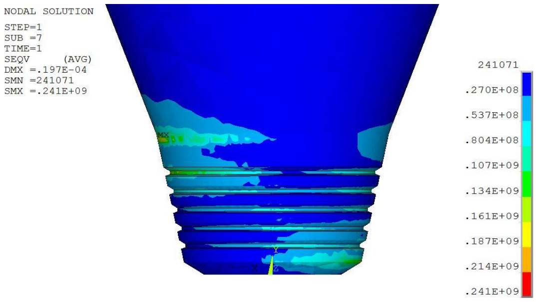
The structural analysis followed the method presented in ISO14801,[12] which is applied to the fatigue testing of dental implant systems. ANSYS Classic[13] was used as the structural analysis tool. The abutment and the fixture consisted of titanium alloy Ti-6A1-4V. Table 1 presents the physical properties of this material[14].

Two loading conditions were applied to the abutment: a chewing force, which acted as an external force, and pretension against the initial clamping force applied to the screw between the abutment and the fixture. Here, the chewing force was a 30° inclination force following the above ISO standard (250 N). The initial tightening torque between the abutment and the fixture was 0.35 Nm, and a pretension of 177.4 N was calculated using the formula[15] for screw clamping force and torque.

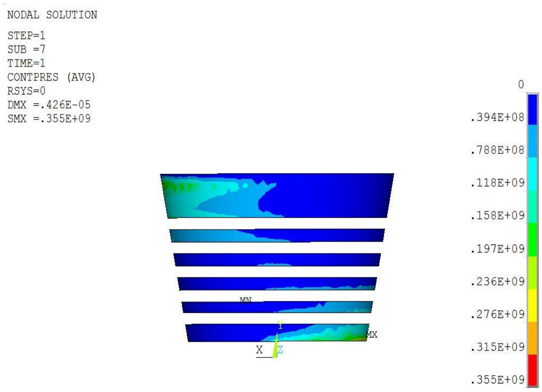
If an external force is applied to the sealing abutment, inter-sliding occurs between the abutment and the fixture. In consideration of this contact, non-linear structural analysis should be conducted. The boundary conditions for the contact between the abutment and the fixture was set to "frictional", and the friction coefficient was set to 0.2. In ANSYS, the abutment was classified as the “contact surface” and the fixture as the “target surface,” while the outside of the fixture was classified as “fixed support.” Finally, because the implant was symmetrical, it was modeled as such, and the “symmetry” condition was applied to the cut section[13].

Table 1 Material property of the titanium alloy

Item	Property
Density [kg/m^3]	4430
Young's modulus [GPa]	113.8
Poisson's ratio	0.342
Compressive yield strength [MPa]	970
Tensile ultimate strength [MPa]	950



(a) Von-mises stress distribution

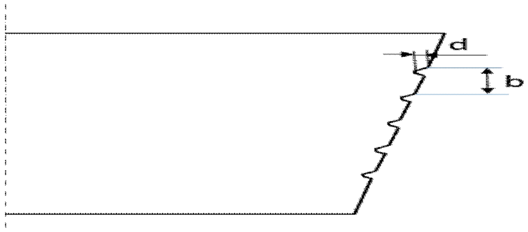


(b) Contact pressure distribution

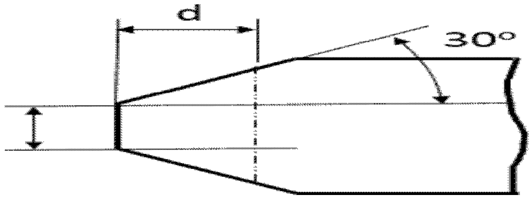
Fig. 3 Results of structural analysis on the existing sealing abutment model

2.2 Structural analysis results

Finite element analysis was conducted on the existing sealing abutment, which was modeled prior to optimization based on the previously set boundary and loading conditions. The results are presented in Fig. 3. The stress distribution (Fig. 3a) indicates that maximum stress occurred in the uppermost area of the contact between the abutment and the fixture. The maximum stress was 241 MPa, and the safety factor was 4.0, which indicated that the strain occurred within the elastic section. The contact pressure distribution (Fig. 3b) revealed that the maximum contact pressure (355 MPa) occurred in the lowermost area of the contact between the abutment and the fixture.



(a) Half cross-sectional view of the sealing abutment with five grooves



(b) Configuration of the tool for cutting the grooves

Fig. 4 Configuration of the cutting tool and the grooves of the abutment

3. Optimization of the abutment grooves

3.1 Groove shape and optimization variables

A sealing abutment was fabricated by machining five grooves on the external side of a general abutment at regular distances. Fig. 4(a) shows the cross-section of the sealing abutment including the grooves, and Fig. 4(b) presents the tip of the cutting tool used to create the grooves. This study employed the tool in Fig. 4(b). The gap between the grooves (b) shown in Fig. 4(a) was fixed at 0.3 mm as in the existing product. The groove shape was modified by changing the groove depth d only as marked in the two figures. Groove depth d was set as the optimization variable while the number of grooves was set at five.

3.2 Configuration of the optimization system

A genetic algorithm[16] was used to optimize the shape of the grooves in the sealing abutment. In the genetic algorithm, the optimization variable (in this case, groove depth) is represented by a single entity consisting of multiple strings or bits, and a group is formed by a number of entities. This genetic algorithm allows non-linear complex problems to be solved through the evolution of the group[16]. The optimization algorithm was run using MATLAB to search for the optimal groove depth. MATLAB and ANSYS exchanged data in real time to perform the optimization task via the sharing of text files. Data during the search processes was delivered to ANSYS, which performed structural analysis based on the variable. The maximum stress and maximum contact pressure were calculated, and the data transferred back to MATLAB iteratively.

3.3 Genetic algorithm

To acquire an optimization solution using a genetic algorithm, a fitness function $F(x)$ first needs to be defined. Generally, a fitness function consists of objective and penalty functions. In this study, the objective function $O(x)$ was set to the following equation to ensure it was proportional to the contact pressure $P_{contact}$ between the abutment and the fixture.

$$O(x) = 1 + P_{contact} / \sigma_Y \tag{1}$$

Here, σ_Y is 970 MPa, which is the yield point, and contact pressure $P_{contact}$ is the maximum contact pressure analyzed in ANSYS for the corresponding entity. As presented in the above equation, the objective function value is always larger than 1.

The penalty function $P(x)$ was set as follow considering the yield point σ_Y and safety factor S_f.

$$P(x) = \begin{cases} 1, \text{ if } \sigma_{\max} > \sigma_Y / S_f \\ 0, \ otherwise \end{cases} \tag{2}$$

Here, $\sigma_{\max}$ is the maximum stress value obtained after analyzing the corresponding entity in ANSYS, and the safety factor S_f was set to 5. As presented above, a penalty was applied if the maximum stress of the abutment was larger than the allowable stress (σ_Y / S_f).

Next, the fitness function $F(x)$ was set as follows using the above two equations.

$$F(x) = O(x) - P(x) \tag{3}$$

As shown in the equation, $O(x)$ is always larger than 1, and the maximum value of $P(x)$ is 1. Thus, $F(x)$ was set to be a positive value that was always larger than 0.

Table 2 presents the parameter values for the genetic algorithm used in the optimization. An individual consisted of genes that have 60 binary values. If a population size is too small, it will quickly converge to a local solution. If the population size is too large, the computation time will increase disproportionately compared to the increase in performance. Thus, this study set the population size to 8. In addition, the maximum number of generations, which establishes when the genetic algorithm will terminate, was set to 30 after identifying the convergence trend through simulations. The crossover probability (P_c) and mutation probability (P_m) were also set within generally known ranges (P_c: 80-95% and P_m: 0.01 -1%)[17].

Fig. 5 shows the overall optimization process for the genetic algorithm and structural analysis.

Table 2 Parameters of the GA

Parameters	Valve
Number of binary genes	60
Population size	8
Maximum generation	30
Probability of crossover, P_c	0.8
Probability of mutation, P_m	0.01

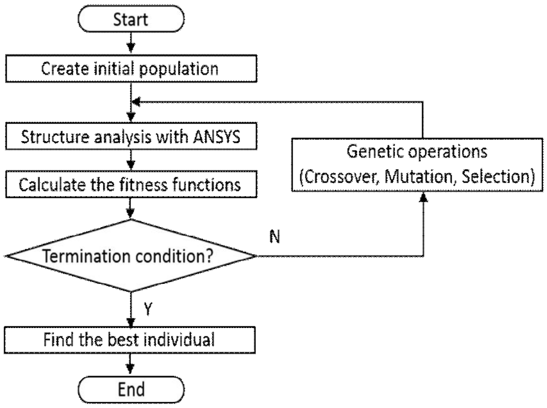


Fig. 5 Flow chart of the proposed genetic algorithm

3.4 Optimization results and analysis

As described above, the optimization was conducted by setting the groove depth d as the optimization variable (Fig. 4a) using the proposed genetic algorithm.

Fig. 6 shows the change in the fitness function by generation. Groove height h was automatically determined according to depth d by the shape of the tool that produces the groove (Fig. 4b). As shown in Fig. 6, the fitness function tended to increase with the number of generations. Because no change was shown after 11 generations, this was regarded as the optimal solution.

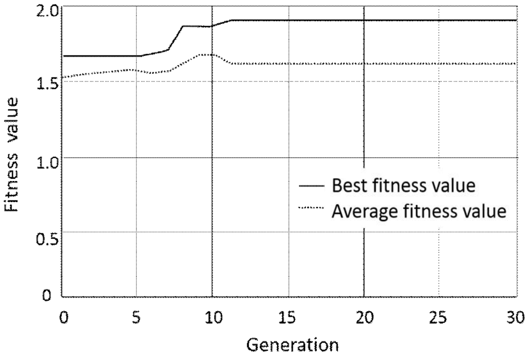
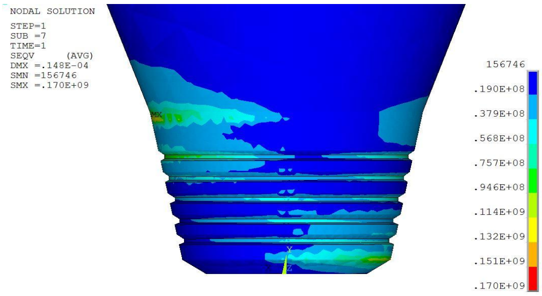


Fig. 6 Change of the fitness function according to generation

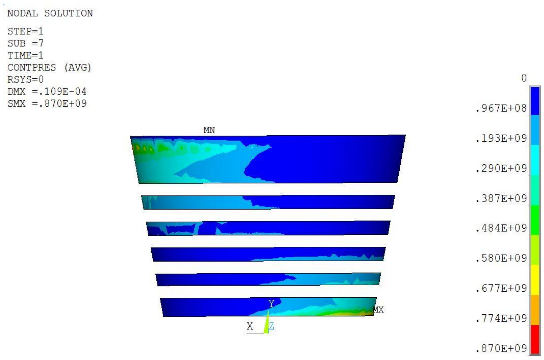
The groove depth d that corresponded to the optimum solution was found to be 0.031 mm, a reduction of 0.019 mm from the existing model (0.05 mm). Accordingly, the groove height was slightly educed as well, and the protrusion area in the abutment increased, thereby making the contact surface slightly larger.

The results of the structural analysis for the optimal solution are shown in Fig. 7. The maximum stress of the optimization model was 170 MPa, a reduction of 71 MPa (29.5%) compared to the initial model (241 MPa). The maximum contact pressure was 870 MPa, an increase of 515 MPa (145.0%) compared to the initial model (355 MPa). The above results are summarized in Table 3.

As presented in the summary, the maximum stress was reduced by approximately 30% compared to that of the existing model under the same xternal force while the contact pressure increased 1.5 times. This result indicates that the sealing performance was improved by the optimization model.



(a) Five-groove von-mises stress



(b) Five-groove contact pressure

Fig. 7 Structural analysis on the optimized model

Table 3 Comparison of the optimized model with the initial model

	Inital (A)	Optimized (B)	(B)-(A)	((B)-(A))/ (A)×100%
Max. stress [MPa]	241	170	-71	-29.5
Max. contact pressure[MPa]	335	870	515	145.0

4. Conclusions

This study performed structural analysis and dimension optimization of five grooves on the outer surface of a sealing abutment using a genetic algorithm to improve the sealing performance abutment and the fixture in a dental implant.

The number of grooves and the gap between the grooves were set to be the same as an existing product, while the groove depth was set as the optimization variable. Based on the results of the optimization process, the performance of the proposed design and that of the existing product were compared using structural analysis. The optimization results showed that the optimal groove depth was slightly smaller than that of the existing model, which reduced stress by approximately 30% when an external force was applied, while the maximum contact pressure, which is directly related to sealing performance, increased by 145%. These results indicate that the optimized sealing abutment model improved sealing performance over the existing model.

Acknowledgments

This research is financially supported by Changwon National University in 2017~2018.

REFERENCES

1. Brånemark, P. I., Breine, U., Adell,, R. Hansson, B. O., Lindstrom, J. and Olsson, A., “Intraosseous Anchorage of Dental Prostheses. I. Experimental Studies”, Scand J. Plast Reconstr Surg., Vol. 3, pp.81-100, 1969.
2. Carlson B. and Carlsson G., "Prosthodontic Complications in Osseointegrated Dental Implant Treatment,” Int. J. Oral and Maxillofacial Implants, Vol. 9, No. 1, pp.90, 1994.
3. Kim, K, N, Choi, K, H. and Oh, H, S., “The evaluation of Fitness on Implant Bolt Joints as Tightening Torques,” The Korean Society of Mechanical Engineers, Vol. 34, No. 11, pp.1786-1791, 2010.
4. Kwon, Y. J., “Structural Design of a Dental implant (I) : Comparative Structural Analysis,” Journal of Computation Structural Engineering Institute of Korea, Vol. 25, No. 5 pp. 421-432, 2012.
5. Siamos, G., Winkler S. and Boberick, K.G., “The Relationship between Implant Preload and Screw Loosening on Implant-Supported Prostheses,” Journal of Oral Implantology, Vol. 28, No. 2, pp.67-73, 2002.
6. Haack, J. E., Sakaguchi, R. L., Sun, T. and Coffey J.P., "Elongation and Preload Stress in Dental Implant Abutment Screws," The International journal of oral & maxillofacial implants, Vol. 10, No. 5, pp. 529-536, 1995.
7. Martin, W. C., Woody, R. D., Miller, B. H. and Miller, A.W., “Implant Abutment Screw Rotations and Preloads for Four Different screw Materials and Surfaces”, The Journal of Prosthetic Dentistry, Vol. 86, No. 1, pp.24-32, 2001.
8. Lee, M. Y., Heo, S. J., Park, E. J. and Park, J. M., “Comparative Study on Stress Distribution around Internal Tapered Connection Implants according to Fit of Cement-and-Screw-Retained Prosthesis,” The Journal of Advanced Prosthodontics, Vol. 5, No. 3, pp.312-318, 2013.
9. Kim, N., Choi, K. H. and Oh, H, S., “The Evaluation of Fitness on Implant Bolt Joints as Tightening Torques,” Proc. of The Korean Society of Mechanical Engineers of Autumn Conference, pp. 1786-1791, 2010.
10. Kim, J. M. and Hong, D. S., "Development of a Sealing-Type Abutment for Implant and the Performance Evaluation via Structural Analysis," J. of the Korean Society for Precision Engineering, Vol. 33, No. 9. pp.769-775, 2016.
11. Hong, D. S., Jeon, Y. J., Lee, H. Y. and Lee, S. Y., "Development and Performance Evaluation of a Sealing-Type Abutment Inserted with a Gold Ring," J. of the Korean Society of Manuf. Process Engineers, Vol. 15, No. 2. pp.97-103, 2016.
12. ISO (International Organization for Standardization), ISO 14801 : Dentistry-Implants-Dynamics Fatigue Test for Endosseous Dental Implants, pp.1-16, 2016.
13. ANSYS Inc., ANSYS Mechanical APDL Structural Analysis Guide, TEA SUNG S&E Inc, 2013.
14. Boyer, R., Welsch, G. and Collings, E. W. eds, Materials Properties Handbook: Titanium Alloys, ASM International, Materials Park, 1994.
15. Hong, J. P., Mechanical Design, BooksHill, pp.100-106, 2001.
16. Goldberg, D. E., Genetic algorithms in search, optimization, and machine learning, Addison-Wesley, 1989.
17. Jin, G. K., Genetic Algorithms and its Applications, Kyowoo Publication, pp. 113-194, 2002.