Reduction of the tooth-implant components dimensions by optimization procedure*

Krzysztof Szajek, Marcin Wierszycki and Tomasz Łodygowski

Institute of Structural Engineering, Poznań University of Technology
ul. Piotrowo 5, 60-965 Poznań, Poland
e-mail: krzysztof.szajek@put.poznan.pl

Restorations with the application of implants are effective and commonly used in dental treatment. The increasing demand of using them in more biological complicated cases frequently induces the needs of small diameter implants. However, many problems connected with this type of restoration are reported. Although the most serious, from patient safety point, is fatigue fracture, the reduced diameter affects also screw loosening and the implant component micromotions. The paper presents the optimization procedure of diameter minimization of commonly used two-component implantology system. The approach employs hybrid algorithm which uses genetic and Hooke-Jevees algorithm. The fatigue life, screw loosening/tightening moment and micromotions level are estimated basing on FE analysis.

Keywords: dental implants, optimization, diameter reduction, hybrid algorithm, genetic algorithm, Hooke-Jeeves algorithm.

1. Introduction

Small diameter implants are frequently used in areas where prosthetic space is limited. Reliable implants with reduced diameter increase a number of cases where dental restoration can be applied. Therefore, all producers make an effort in order to introduce them to the offer. Yet, the problems with small diameter implants usage are frequently reported in clinical restorative study [7]. The problems concern: fatigue fracture [3,6], screw loosening [5] and implant-bone interface overloading [4]. In consequence, the offer of small diameter implants is limited in the market and practitioners should take into account biomechanical risk before its application. For the sake of above it is important to develop the procedure which allows for the implant diameter minimization without worsen the rest characteristics beyond acceptable limits.

2. Model

The two-component implantological system is considered (OSTEOPLANT). It consists of root and abutment which are connected with a non-rotational hexagonal slot, assembled by a screw (Fig. 1a). The geometry of implant may be simplified to axisymmetric but the loads and response are asymmetric. However, the axisymmetric geometry enables us to use a special type of finite elements, which use standard isoparametric interpolation in the radial – symmetry axis plane, combined with the trigonometric interpolation functions with respect to the angle of revolution [1]. This formulation enables us to describe nonlinear asymmetric deformation of axisymmetric geometry due to asymmetric loads and simultaneously significantly reduces the size of the problem (ca. 94 000 dof) in comparison with a full three dimensional model (ca. 600 000 dof).

All components of the implant are made of medical alloys of titanium. In the FE model the isotropic, non-linear elastic-plastic characteristics of material models are taken into account. The whole implant parts are positioned relative to each other. Fully relative position constraints of implant parts make possible automatic redefinition of implant model assembly. When the geometry of the parts of implant model is modified during optimization procedure the assembly is updated automatically. The simulation of assembled structure involves solving a contact problem. In the case of two-component implant system it is necessary to define three contact areas between: root and abutment, root and screw, abutment and screw. The penalty method is used as the contact constraint enforcement method for both normal and tangential behavior [8]. Tangential surface behavior has been defined as classical isotropic Coulomb friction model. The friction coefficient is the same for all contact pairs and amounts to 0.19.

The loads are applied in two steps. The first step is simulation of tightening. The pre-tension force is defined in the middle part of the screw. The value of this force in a tightened screw was calculated from the analitical equation [2]. The second step is bending, which is caused by the worst component of service load, perpendicular to axisymmetric axis. The bending force (20N) is applied to the tip of abutment by means of a surface-based coupling constrain.

3. Model parametrization

The geometry of presented two component implantology system is quite complex so the choice of correct design parameters is crucial. Seven geometrical parameters and screw preload were defined as real variables (Fig. 1a). All geometric parameters refer to the upper part of the implant. The thread was excluded from the consideration at this stage of the work.

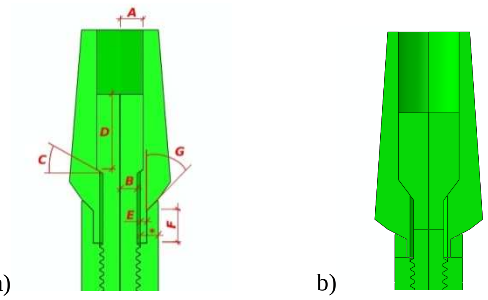
For each design parameters range and, due to GA requirements, number of bit for encoding were defined. The ranges come from both, the geometry limitations and manufacture requirements.

In automatic optimization process the reliability of building FE model on the basis of design parameters vector is crucial. Any error disturbs the design space and makes the optimization more difficult. In order to assess the reliability a series of test for randomly generated design parameters were calculated. The average error (unphysical geometry was not treated as an error) equals 17% and in spite of many trials it was not reduced.

* For this research the financial support was provided by the grant N R13 0020 06. This help is kindly ackonowledged.

4. Optimization procedure

The goal of presented optimization was to minimized the diameter of the dental implant with the subject to constrains: fatigue life, more than 1mln cycles; the loosening moment higher than 20Ncm; tightening moment lower than 35Ncm and no relative displacement of contacting surfaces in the implant body-abutment and abutment-screw interfaces. The static, exterior penalty approach is used in order to control the constraints.



a) b)

Figure 1: Dental implant geometry: a) the initial model with assumed geometric design parameters, b) the best solution obtained.

The optimization bases on complex, nonlinear FE model, which reliability does not ensure the result for all configurations of design parameters. Therefore, genetic algorithm (GA) which is resistant to non-continuous design space was chosen. Unlike conventional optimization techniques GAs explore simultaneously the entire design space and therefore are likely to reach the global optimum. Unfortunately, in the same time the algorithm is computationally expensive, especially if a precise solution is expected. As a result, the hybrid method is proposed in order to make both, search the design space globally and obtain the precise solution. The presented approach starts from genetic processing and after a few iterations the best obtained solution is assumed to be the starting point of Hooke-Jevees procedure which stops when sufficient accuracy of the solution is achieved.

Evaluation of each proposed design bases on FE analysis carried out with Abaqus. The chromosomes created during genetic processing or directly design parameters in case of Hooke-Jevess approach constitute the starting point for evaluation procedure. For each design parameters configuration, FE model is created and simulation is performed. The obtained results are interpreted according to the given objective function.

5. Results

The optimization procedure was carried out three times and the results obtained provided the same solutions. An example ran of the presented approach as an objective function evaluation in dependence to FE analysis number is shown in Figure 2 while the best solution obtained is presented in Figure 1b. The diameter in the optimal solution was reduced to 0.64mm (17.5% reduction) what resulted in the implant body diameter equalled 3.1mm (3.75mm in the initial model). None of the defined constraints was violated and equalled as follows: fatigue, 6.3mln of cycles; the screw loosening moment, 22Ncm; tightening moment, 26.3Ncm; no interfaces opening.

The screw preload value for the optimal solution was established on the level of 560N (425N in the initial model). It is frequently reported that the increase in screw preload results in screw fatigue life reduction [6]; however, in the presented solution due to simultaneous change in geometry, higher preload caused compressive stresses increase so that fatigue resistance is not reduced below assumed limit.

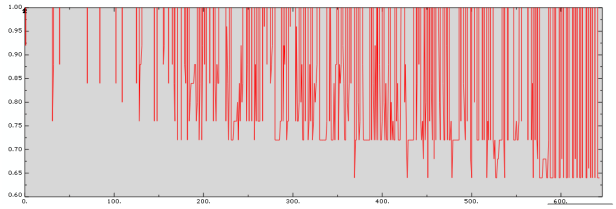


Figure 2: Objective function evaluation in dependence to FE analysis number

6. Conclusions

The complex dental implant model was optimized based on FE analyses which was successfully incorporated into the hybrid optimization procedure (genetic and Hooke-Jeeves algorithms).

The study and the final results give evidence that presented method is efficient and can be used for dental implant diameter reduction.

The obtained proposition of new design has reduced diameter without violating the defined constraints (fatigue life, loosening and tightening moment, interfaces separation).

Although the design needs to be adjusted to manufacturer's possibility and tested carefully, it can be seen as a direction for further modification in the existing implantology system.

References

[1] *Abaqus Analysis User's Manual*, SIMULIA., Providence, 2010.

[2] Bozkaya, D., Muftu, S., Mechanics of the taper integrated screwed-in (TIS) abutments used in dental implants, *J. Prosthet. Dent.*, 91, pp. 326-334, 2004.

[3] Flanagan, D., Ilies, H. McCullough, P. and McQuoid, S., Measurement of the fatigue life of mini dental implants: a pilot study, *J. Oral. Implant.*, 34, pp. 7-11, 2008.

[4] Himmlova, L., Dostalova, T. Kacovsky, A. and Konvickova, S., Influence of implant length and diameter on stress distribution: A finite element study, *J. Prosthet. Dent.*, 91, pp. 20-25, 2004.

[5] Hoyer, S.A., Stanford, C.M. Burandadhom, S., Fridrich, T., Wagner, J. and Gratton, D., Dynamic fatigue properties of the dental implant-abuatment interface: Joint opening in wide-diameter versus standard-diameter hex-type implants, *J. Prosthet. Dent.*, 85, pp. 599-607, 2001.

[6] Łodygowski, T., Wierszycki, M., Szajek, K., Hędzelek, W., Zagalak, R., *"Tooth-Implant Life Cycle Design", Computer Methods in Mechanics, Lectures of the CMM 2009, ASM 1, chapter 21*, pp. 391-422, Springer-Varlag Berlin Heidelberg 2010.

[7] Rangert, B., Krogh, P.H. Langer, B. and Van Roekel, N., Bending overload and implant fracture: a retrospective clinical analysis, *Int. J. Oral. Max. Impl.*, 10, pp. 326-334, 1995.

[8] Wriggers, P., *Computational Contact Mechanics*, John Wiley and Sons, 2002.