Acta of Bioengineering and Biomechanics
Vol. 18, No. 1, 2016

Original paper
DOI: 10.5277/ABB-00323-2015-02

Numerical verification of two-component dental implant in the context of fatigue life for various load cases

KRZYSZTOF SZAJEK*, MARCIN WIERSZYCKI

Poznań University of Technology, Institute of Structural Engineering, Poznań, Poland.

Purpose: Dental implant designing is a complex process which considers many limitations both biological and mechanical in nature. In earlier studies, a complete procedure for improvement of two-component dental implant was proposed. However, the optimization tasks carried out required assumption on representative load case, which raised doubts on optimality for the other load cases. This paper deals with verification of the optimal design in context of fatigue life and its main goal is to answer the question if the assumed load scenario (solely horizontal occlusal load) leads to the design which is also "safe" for oblique occlussal loads regardless the angle from an implant axis. *Methods*: The verification is carried out with series of finite element analyses for wide spectrum of physiologically justified loads. The design of experiment methodology with full factorial technique is utilized. All computations are done in Abaqus suite. *Results*: The maximal Mises stress and normalized effective stress amplitude for various load cases are discussed and compared with the assumed "safe" limit (equivalent of fatigue life for 5e6 cycles). *Conclusions*: The obtained results proof that coronial-appical load component should be taken into consideration in the two component dental implant when fatigue life is optimized. However, its influence in the analyzed case is small and does not change the fact that the fatigue life improvement is observed for all components within whole range of analyzed loads.

Key words: *dental implant, optimization, fatigue life, genetic algorithm*

1. Introduction

Nowadays, dental implants are common way of treatment in the case of edentulous patients. However, in spite of high rate of success there are still a few problems observed. Among other things, one of the most important, from patient's safety point of view, is implant fracture due to both the static overloading as well as fatigue damage. While capability for static load can be relatively easily estimated the fatigue life prediction is still the subject of many investigations.

A systematic review of twenty-six follow-up studies estimated a cumulative incidence of implant fractures after 5 years at a level of 0.14% [12]. However, the fracture ratio for the follow-up studies for up to 15 years [1] rises drastically and can reach even 16% in the maxilla. This proves both fatigue fracture nature of recorded failure and weakness of follow-up studies limited to 5 years in the context of sufficient dental implant fatigue life confirmation.

Fatigue fracture is a complex process strongly depending on mechanical and biomechanical factors such as marginal bone loss and screw stability [10], [19]. Many approaches considering dental implant life estimation are present in the literature based on laboratory tests [5]–[7], [11], [20], 21] and numerical approach, mostly finite element method [8], [9], [24]. A review of them allows us to conclude that as regards implant fatigue life the most crucial factors are implant–abutment connection geometry, a screw preload, dental implant fixation and crown loading.

In order to meet the need of dental implant design tool, a complete procedure for improvement of two-component dental implant was proposed [22]. The

---

* Corresponding author: Krzysztof Szajek, Institute of Structural Engineering, Poznań University of Technology, ul. Piotrowo 5, 60-965 Poznań, Poland. Phone number: +48 61 6652103, e-mail: krzysztof.szajek@put.poznan.pl

Received: May 15th, 2015
Accepted for publication: May 26th, 2015

procedure is based on genetic algorithm hybridized with Hooke–Jeeves technique. The objective function is estimated through finite element analysis with simplified model, yet, capable to estimate all necessary dental implant features. The optimization problem formulation in the context of fatigue life is proposed as finding a configuration of geometry parameters for which the optimal screw preload maximizes the number of cycles to fatigue. Static failure risk and adequate levels of screw loosening/tightening moments were constrained. An optimal solution with meaningful fatigue life improvement and meeting the assumed constraints was proposed. The elementary verification considering influence of FE model simplification has been done [22] and revealed small quantitative differences; yet, the correctness of qualitative measure has been proved.

A few questions arise in the context of formulated optimization problem and optimal design obtained. The most important considers assumed load scenario and boundary conditions. Load scenario, in order to eliminate individual variability, should represent a simple but the worst possible case. In the literature, the oblique direction is frequently proposed [11], [13] which is biologically justified and causes superimposition of bending and compression in the joint component [15]. However, the problem is what should be the load angle from an implant axis. The bucco-lingual (horizontal) component has clear effect on fatigue life. It causes extreme tension in the screw [2], which usually leads to the minimal fatigue life. It was also confirmed by Genna [8], [9] for low-cycle fatigue failure. Therefore, this load component was assumed solely in the optimization problem [22]. However, an influence of coronial-apical (vertical) component cannot be easily predicted. On the one hand, it can lead to the plastic deformation on the fixture-abutment interface and circumferential region of the fixture [23]. On the other hand, it reduces tensile stresses generated due to bending [2] and preload, which decreases both the mean stress and its amplitude. Thus, in the context of screw fatigue life, coronial-apical load component seems to make it higher. This leads to the conclusion that as long as screw is the weakest element, the assumption of only coronial-apical load component represent the worst case. Regardless of right selection of the most adverse case, unfortunately, there is still no guarantee that the final optimal solution is still "optimal" for other component combinations. Another problem to be verified is the influence of load magnitude on optimal solution. A dental implant is a nonlinear system and the region most jeopardized to a fatigue fracture can change place depending on the load magnitude. Even though, it is possible to meet elastic regime constraints in the optimization, possible relative movement on contacting surfaces between all implant components can change failure mode. Additionally, one should also note that the optimization problem formulation did not include restriction of sensitivity of the fatigue life to load magnitude. It was assumed that the choice of the worst load component leads to the "safe" design also for slightly different load magnitudes, especially lower ones. All these doubts give rise to the question of whether the optimal design obtained for load with fixed angle and magnitude is also optimal or at least "safe" for other loads. The answer is especially important when the implant is expected to be used in a wide spectrum of applications and various tooth replacement. Therefore, in this study, the optimal implant fatigue life for various load component combinations and their magnitudes is verified. The results are also compared with the initial design to see the improvement achieved.

The numerical verification utilizes design of experiment scheme with a full factorial pattern. Only two parameters are introduced, vertical and horizontal component magnitude. The analyses were done using Abaqus suite. The fatigue life estimation utilizes stress based approach with linear Goodman theory.

2. Materials and methods

2.1. Optimization definition and result

A detailed description of the problem and optimization strategy can be found in [22]. Because some elements are common to computational model used in this study and for the sake of verification clarity, below the most important information is recalled.

An example of two-component implantological system is considered (Fig. 1a). The geometry is simplified to axisymmetric one, however, the asymmetric deformations are considered. The components are made with titanium or its alloys modeled as non-linear elastic-plastic materials. Between all components contact conditions are defined – "hard" Hertz contact in normal direction and isotropic Coulomb friction model in tangential direction. A two step procedure is assumed, simulation of tightening (425 N) and bending (load of 30 N perpendicular to axisymmetric axis, tip of the abutment). The implant root is fixed assuming moderate three millimeter bone loss.

The fatigue life estimation in the optimization as well as in the following verification uses stress based approach with Goodman linear theory where the effective stress amplitude at zero mean stress is calculated as follows

$$\sigma_{ar} = \frac{K_f \overline{\sigma_a}}{1 - \frac{\overline{\sigma_m}}{\sigma_0}} . \tag{1}$$

In the above, the effective amplitude of stresses, $\overline{\sigma_a}$, is calculated on the basis of Mises stress, effective mean value of stresses, $\overline{\sigma_m}$, is given as a first invariant of stress tensor, while K_f is the fatigue-reduction factor (for fixture 1.25 – strongly roughed surface). Assuming constant values of stress amplitude and stress mean value, the number of maximal cycles to fatigue failure can be given by

$$N = \frac{1}{2}(\sigma'_{ar})^{\frac{1}{b}}, \quad \sigma'_{ar} = \frac{\sigma_{ar}}{\sigma'_f} \tag{2}$$

where σ'_f is fatigue strength coefficient. Equation (2) shows that an increase of σ'_{ar} prolongs the fatigue life. Therefore the following problem formulation was proposed

$$\max_{x \in X} \sigma'_{ar}(x) .$$

Subject to:

$$\begin{aligned} &S(x) \le 1.0 , \\ &L(x) \ge 150 \ \text{Nmm}, \\ &T(x) \le 300 \ \text{Nmm}, \end{aligned} \tag{3}$$

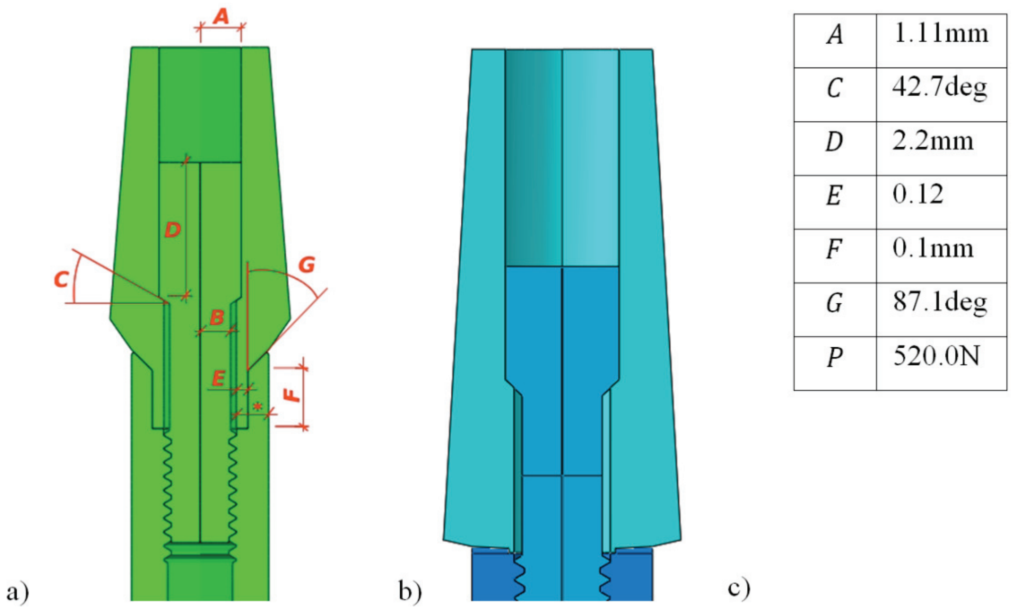
where x denotes seven geometrical parameters (Fig. 1) and screw preload, $S(x) = \frac{\sigma_{\text{Mises}}}{\sigma_{\text{yield}}}$ represents static risk factor, while $L(x)$ and $T(x)$ denote loosening and tightening moments calculated using Bozkaya's formula [3].

A penalty method was used as a constraint-handling technique. Let us assume that $h_{i,\text{viol}}$ is a measure of the i-th constraint violation which equals if constraint is not violated, otherwise $|h_i - h_{i,\text{limit}}|$, where h_i and $h_{i,\text{limit}}$ denote values of constraint function and assumed limit, respectively. The problem expressed in equation (3) can then be transformed into

$$\max \left(\sigma'_{ar}(x) - \sum_{i=1}^{n} k_i \cdot \Phi[h_{i,\text{viol}}(x)] \right) \tag{4}$$

where Φ is the penalty function while k_i denotes penalty coefficient. The following exterior and dynamic penalty function was assumed

$$\Phi[h_{i,\text{viol}}(x)] = (\rho \cdot t)^{\xi} [h_{i,\text{viol}}(x)]^{\beta} \tag{5}$$



A	1.11mm
C	42.7deg
D	2.2mm
E	0.12
F	0.1mm
G	87.1deg
P	520.0N

Fig. 1. Dental implant geometry: (a) the initial model with assumed geometric design parameters (parameter *E* is defined as a fraction of * distance), (b) the best solution, (c) optimal configuration of design parameters

where $\xi = 2.0$, $\beta = 2.0$ and $\rho = 1.0$, while t denotes iteration number. The penalty coefficient is defined as follows

$$k_i = \frac{|\,\sigma'_{ar,\max} - \sigma'_{ar,\min}\,|}{[0.1(h_{i,limit} - h_{i,\mathrm{extreme}})]^2} \tag{6}$$

where $h_{i,\mathrm{extreme}}$ is the estimation of extreme (maximal if constraints create upper bound, otherwise minimal) value of the i-th constraint, $\sigma'_{ar,\max}$ and $\sigma'_{ar,\min}$ denote estimation of maximal and minimal possible value of the objective function (the estimation for $h_{i,\mathrm{extreme}}$, $\sigma'_{ar,\max}$ and $\sigma'_{ar,\min}$ is based on the large number of analysis carried out during the preliminary study and reliability test for various design parameter configurations, while $h_{i,\mathrm{limit}}$ is the assumed limit for the given constraint.

The genetic algorithm hybridized with the Hooke–Jeeves procedure was used to solve the problem. In the first stage a population of sixty random solutions was generated. Each individual represented one design parameter configuration encoded in chromosomes using Gray coding. Next, the population was iteratively subjected to genetic operators such as crossover and mutation. Forming the next population the individual with higher value of objective function was preferred (rank-based selection). After ten iterations the best solution obtained was the start point for Hooke–Jeeves procedure which quickly found the design minimizing the objective function.

The best solution is presented in Fig. 1b. The objective function is reduced by 95% to 0.0197, which is en equivalent of 4.5e17 cycles to fatigue fracture. All constrained values fulfil assumed limits and equal 0.49, 217 Ncm and 285 Ncm for static risk factor, loosening and tightening moment, respectively.

The optimization problem definition due to simplifications ignored the influence of new geometry on change of stresses/strains in bone–implant interface [18]. Therefore, the additional analyses were carried out with full spatial geometry implant loaded with 30 N of horizontal load and fixed in bone-like material (D1, linear, elestic with Young's modulus equal 13 GPa and 9.5 for cortical and cancellous bone, respectively) with and without 3 mm marginal bone loss. The results showed the stresses/strains concentration in marginal regions for all the cases, which is in agreement with other studies [25]. The optimal design leads to stress/strain reduction of 25% model without marginal bone loss, which proves the risk reduction of bone overloading. No stress/strains change was observed when marginal bone loss was included.

2.2. Finite element analyses

Fully spatial FE model with the same definition of material and contact conditions as in the axisymmetric one (used in optimization) has been used for verification. The geometry follows the geometric parameters configuration for the initial and optimized design. However, helix thread for both fixture and screw, hexagonal slot (fixture–abutment connection) and hexagonal mortise are considered. Additionally, the implant root is fixed in bone-like material (linear elastic, E = 9.5 GPa, D1 according to Lekholm and Zarb index). The FE model consists of 155 K 8-node linear brick elements and 200 K nodes (556 K DOF). The analysis was divided into two steps – screw prestress and the abutment loading with occlussal loads (vertical and horizontal loads are applied simultaneously). Fatigue life is calculated according to the rules given in 2.

A series of 40 numerical simulations were prepared for various magnitudes of occlusal load components and for both the initial and the optimized design. The horizontal load varied from 0 to 90 N with step of 30 N, while the vertical one from 0 N to 400 N with step of 100 N. The screw preload of 425 N and 520 N is applied for the initial and the optimized design, respectively. Due to the small number of variables, full factorial technique of design of experiment was chosen. The computations were carried out using Abaqus/Standard v.6.11 and Fujitsu-Siemens PRIMERGY RX300 S6 (Intel Xeon X5680 6C/12T 3.33 GHz, RAM 8 GB, OS GNU Linux). The computations were automated with home-made Python script using Abaqus Scripting Interface.

3. Results

The maximal Mises stress and normalized effective stress amplitude (see 2) are presented in Fig. 2 and Fig. 3 for the initial design and the optimized one, respectively. In order to make the interpretation easier, Fig. 2 presents contour plots of cycles to fatigue failure (see 2) depending on occlusal load magnitude/angle. The isolines were calculated using equation (2) transformed to the form of $\sigma'_{ar} = (2N)^b$. The limit of 5e6 cycles is indicated with thick isoline.

Table 1. Detail results for particular components in various load scenarios for the optimized design

Load H/V	Abutment				Screw				Fixture			
	σ_m	σ_a	σ_{ar}	N	σ_m	σ_a	σ_{ar}	N	σ_m	σ_a	σ_{ar}	N
30/0	–33.6	24.1	23.1	1.3e19	731.7	6.8	56.0	1.7e15	–40.6	60.3	70.7	6.2e12
30/400	165.3	40.9	51.5	2.9e15	706.5	28.5	188.9	4.8e9	–3133	131.7	109.1	6.5e10

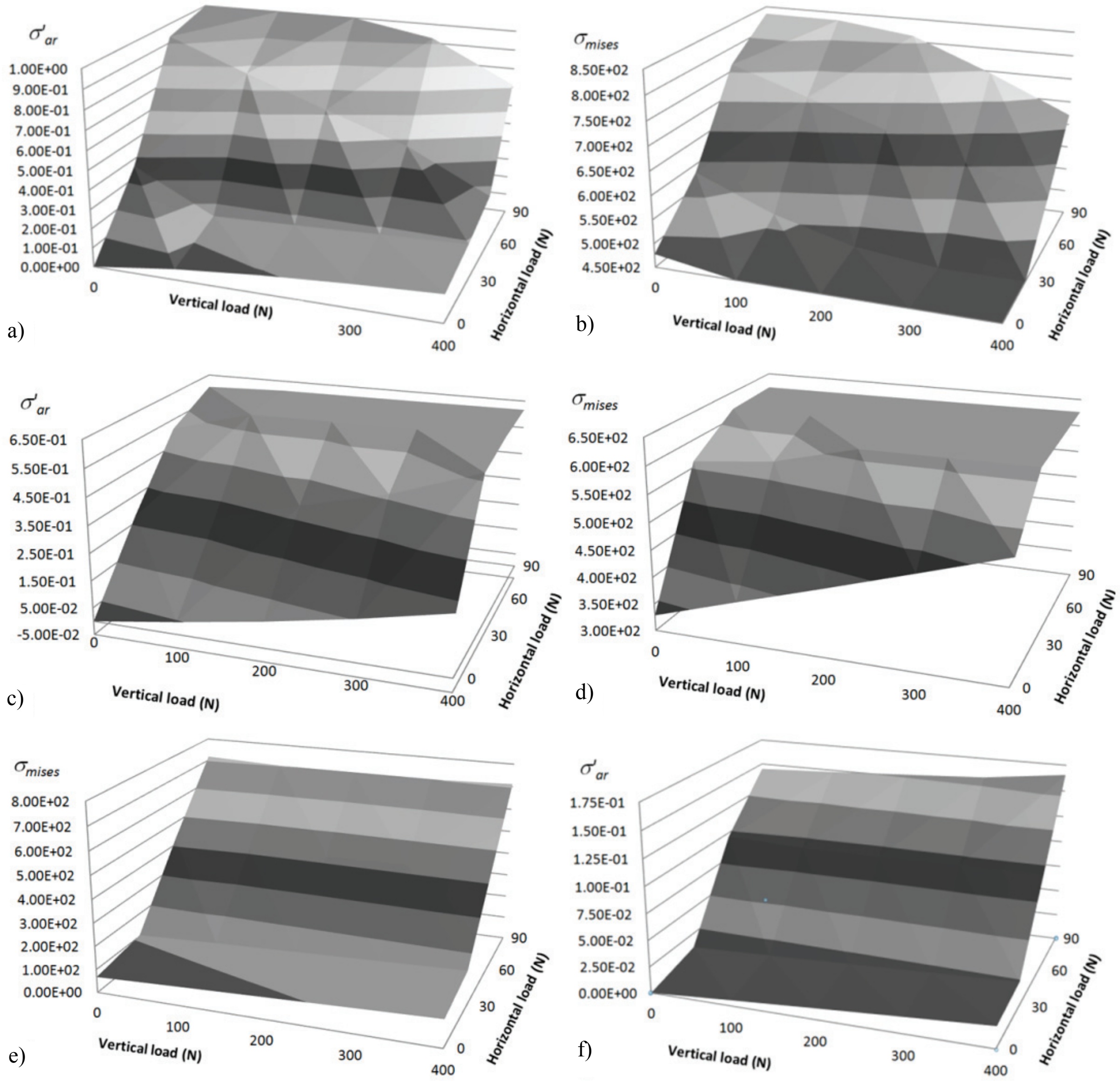


Fig. 2. Initial design. Mises stresses (left column) and normalized effective stress amplitude at zero mean stress (right column) for various combinations of occlusal loads and particular components: (a–b) screw, (c–d) fixture, (e–f) abutment

Dotted areas indicate a region of design where the number of cycles to failure is higher than 5e6. The contour plots are also supplemented with the dotted lines indicating a relation between transversal and vertical components for various angles of loads. Additionally, detail results for fatigue life estimation and various levels of vertical components are listed in Table 1.

The stress field was examined only in exterior nodes excluding contact surfaces and threads as well as internal surface of a fixture (artificial stress concentration in the vicinity of contact surfaces).

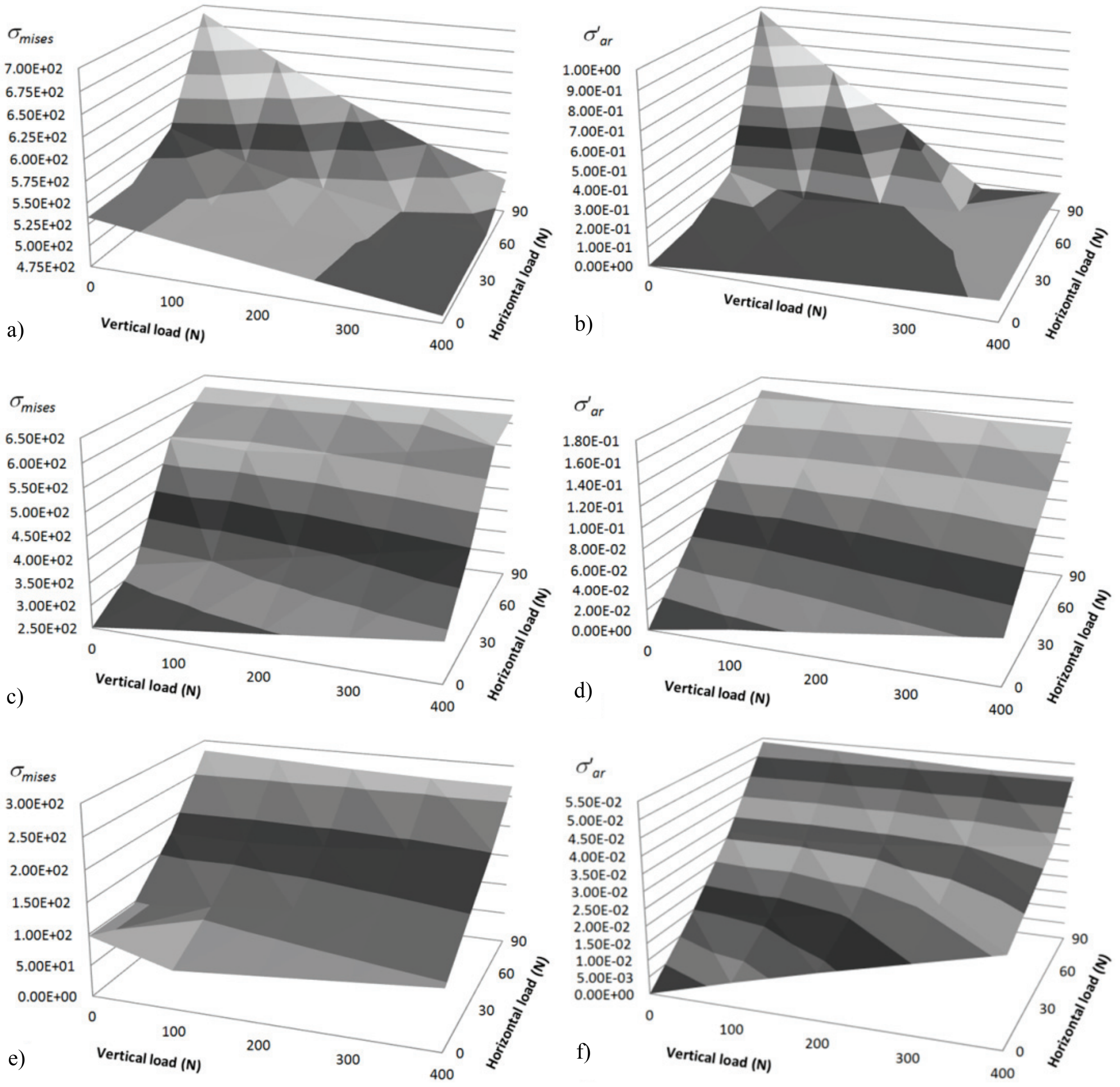


Fig. 3. Optimized design. Mises stresses (left column) and normalized effective stress amplitude at zero mean stress (right column) for various combinations of occlusal loads and particular components: (a–b) screw, (c–d) fixture, (e–f) abutment

4. Discussion

Let us start from recalling the main question set for this study in the introduction, that is "if the optimal design obtained for load with fixed angle and a magnitude is also optimal or at least "safe" for other loads". Proving the optimality for such a complex system requires solving a series of optimization problems for combinations of occlusal loads instead of single analyses and, therefore, it cannot be done based on the obtained results. However, from the practical point of view, it would be enough to prove the optimized implant is "safe" for physiologically justified occlusal loads, which the presented study is limited to.

A further discussion requires formulation of the definition of "safe" solution which appears at the beginning of this section. The optimization done considers screw loosening/tightening limitations themselves and due to the fact that they do not depend on occlusal loads it can be assumed they are fulfilled. Static failure directly depends on Mises stresses which are also included in the formula for fatigue life estimation and therefore there is no need to control it separately. Finally, within the limitations of this study it is enough

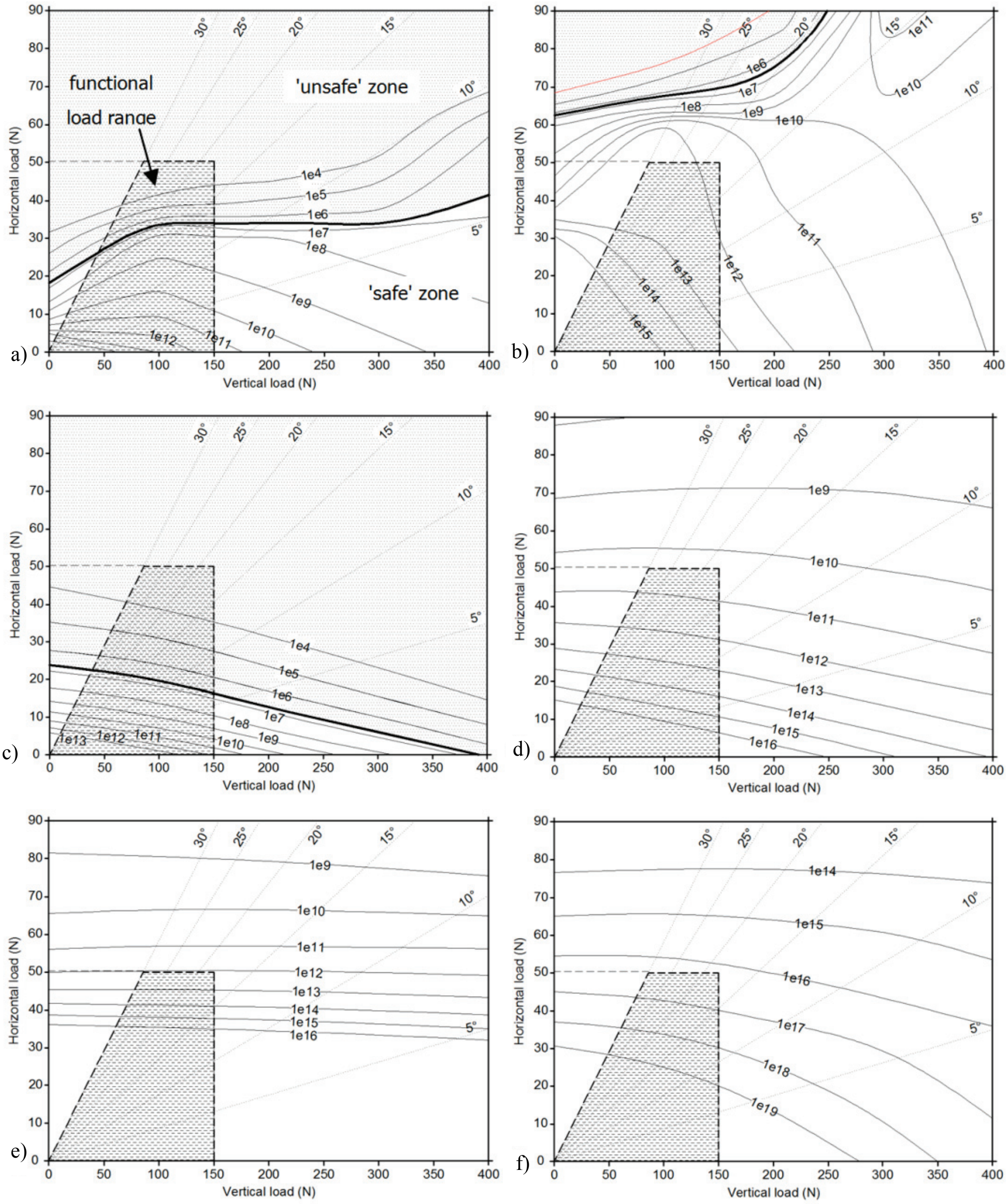


Fig. 4. Contour plot of the number of cycles to fatigue, *N*, for particular components and various combinations of occlusal loads: (a) screw (initial), (b) screw (optimized), (c) fixture (initial), (d) fixture (optimized), (e) abutment (initial), (f) abutment (optimized)

to rely solely on fatigue life. Following the proposition in the standard ISO 14801:2007 the limit of 5e6 cycles to failure was assumed as a limit for “safe” design. It correspond with normalized stress amplitude equal $\sigma'_{ar} = (2N)^b = (5\cdot 1e6)^{-0.095} = 0.231$. However, it should be pointed out that the utilized stress based fatigue life estimation is a rather rough measure and, as a consequence, the assumed level of cycles should be considered as a qualitative measure which can be used for comparison but very carefully as absolute measure.

Another important thing to determine is a set of possible, physiologically justified, load scenarios. According to in vivo force measurement in implants supporting different types of superstructure done by Mericske-Stern et al. [14] the maximal functional forces during chewing food (bread and apple) equal 50 N and 150 N for horizontal and vertical components. Similar values of vertical masticatory forces are reported by Morneburg et al. [16], [17]. The maximal values were mostly registered during the first cycle of chewing and it can be assumed that the frequent value, although hard to estimate, is much lower. As there is no information whether these maximal values can occur simultaneously, in this study the most adverse case is assumed. Therefore, the possible area of functional loads would be represented with a rectangular region. However, taking into account that although load angles from implant axis higher than 30 degrees can occur, but are not frequent, the upper left corner of the triangle is cut off as indicated in Fig. 4. The assumption can also be justified with the guideline of the standard ISO 14801:2007 where the angle of 30 degrees is also proposed for test set-up. The determined region is called "functional load range" further on.

Each component: screw, fixture and abutment is discussed independently. Because of the assumption made in the optimization that the "worst" load case is represented by solely horizontal load of 30 N, an influence of vertical component introduction is analyzed first. Then, viewed from wider perspective, the whole range of assumed magnitudes of particular components is presented based on contour plots in Fig. 4. The discussion covers both the "safety" of the optimized design as well as its improvement in comparison with the initial one.

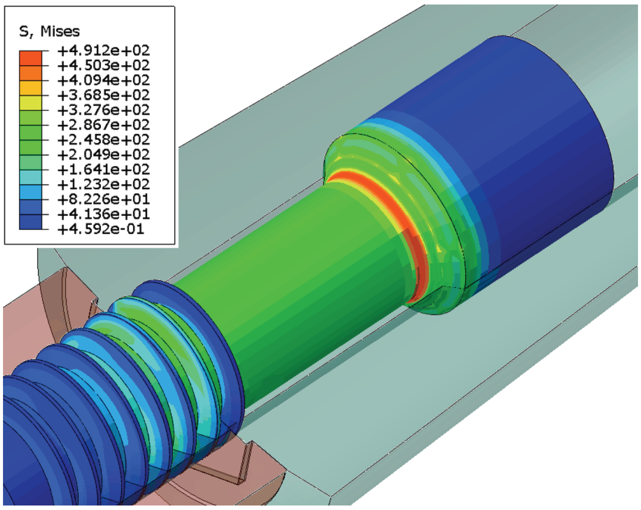


Fig. 5. Mises stresses in a screw of optimized dental implant for horizontal load of 30 N. Red area indicates stress concentration

Let us start the discussion with a screw which is the weakest element in the context of fatigue fracture in the initial design. The maximal stresses for the screw are localized at the transition point from a shank and screw head. This is also the place where fatigue cracks are the most likely to occur (Fig. 5). The fatigue and static mechanism of failure is an effect of screw bending which was rightly modeled in the optimization with solely horizontal load. Let us discuss an effect of vertical load. The abutment undergoing compressive load is pushed into the fixture and causes screw preload reduction. The range of reduction if closely connected with vertical stiffness of an implant [4]. Because the optimal solution is characterized by horizontal interface between abutment and fixture, which increases vertical stiffness significantly, in the case of 30 N horizontal load, the introduction of vertical load of 200 N (even more than defined functional load range) reduces Mises stresses by only 25 MPa (less than 5%) from 534 MPa down to 509 MPa. The reduction follows the common intuition that more prestressed implant is more resistant to bending. Unfortunately, there is no such obvious relation in the case of fatigue fracture. In stress based approach (see equation (1)) a number of fatigue cycles is a function of an average value of the first invariant of stress tensor (effective mean value, σ_m) and effective stresses (amplitude of Mises stress during a cycle, σ_a). Table 1 presents both values for load combinations (horizontal/vertical) 30 N/0 N and 30 N/400 N. It can be seen that vertical load reduces effective mean value but increases effective stresses at the same time. As a consequence, the effective amplitude at zero mean stress increases (the number of cycles to fatigue decreases) when vertical load is introduced and, additionally, the screw becomes the weakest component (the lowest number of cycles to fatigue). Thus, the decrease of fatigue life is caused by decreasing Mises stresses in compressed implant. This effect is clearly observed for the case without horizontal load (Fig. 3b). The maximal Mises stresses are reduced from 533 MPa to 483 MPa when vertical load of 400 N is applied, however, the normalized effective amplitude increases from 0.036 to 0.106 (equivalently N = 9.1e9). The effect of fatigue life decreasing due to the higher value of vertical load is also observed for horizontal load of 60 N. Yet, vertical load raising up to 100 N firstly provides stabilization which prevents screw bending effect. As long as the reduction does not cause the stresses to be lower than the value obtained for pure tightening, the fatigue life increases. For vertical load higher than 100 N, the Mises stress decreases below this limit and effective stress amplitude increases due to the previously described reason. A similar effect can be noticed for 90 N of horizontal load but in this case 300 N of vertical load is necessary to provide abutment stabilization.

Based on the obtained results and utilizing the fatigue life estimation approach, it can be concluded that the assumption that solely transversal load can represent the worst load case for a screw is not correct. The observed fatigue life reduction due to vertical load introduction is an effect of screw preload reduction during a load cycle. In the shakedown analysis done by Genna an effect of screw tightening is incorporated in indirect way on the basis of additional analyses [9]. Therefore, the preload reduction mentioned could not be covered. However, two important results should be highlighted. Firstly, the fatigue life decrease due to vertical load is caused by stresses reduction, which is generally small. Secondly, even if 400 N and 30 N of vertical and horizontal loads, respectively, are assumed, the number of cycles to fatigue failure exceeds 9.1e9 which is still above assumed "safe" limit. Taking into account the above, the observed fatigue life change does not negate the high fatigue life of the obtained design.

The screw component fatigue life improvement achieved due to optimization for solely 30 N of transverse load was proved in the previous study [22]. However, the contour plots presented in Fig. 4 allow us to make the assessment from a wider perspective for other load component combinations. Analyzing the results for the initial design firstly, the "safe" value of transverse load can be estimated at the level of 18 N. The contour plot presents the effect of abutment stabilization when the vertical load component is applied and the allowed value of transverse load exceeds 30 N if it acts together with a vertical load of 100 N at least (equivalently, load of 105 N which acts 18 degrees off implant axis). The "safe" limit became almost horizontal for higher value of vertical load. This is a result of increasing amplitude caused by small stiffness of implant in axial direction [2] and, in turn, screw preload reduction in response to compressive load. The limit increases again for vertical load higher than 300 N. It is important to note that the implant is entirely in "safe" zone for the load which is coincident with an implant axis or acts maximally 5 degrees off it. For higher value of load angle the maximal allowed magnitude decreases to 196.2 N, 130.7 and 52,4 N for 10, 15 and 30 degrees, respectively. Although it can be assumed that a number of cycles for load with particular variation from implant axis decrease when the angle increases there is still small "unsafe" region which is inside the functional load range. In the case of predominant loads with angles higher than 15 degrees the screw is jeopardized to fatigue fracture.

From the comparison of "safety" zones it can be easily seen that the crucial weakness of the initial design, low screw fatigue life, was extremely improved. The screw is able to withstand 5e6 cycles for 60 N of solely horizontal load. The implant failure within presented "unsafe" zone is clearly the result of screw bending, yet, the previously described stabilization due compressive loading makes the allowed transverse load higher reaching the assumed upper level of 90 N for 250 N of compressive load. It the case of an optimized implant the "safe" load angle deviation from implant axis reaches 20 degrees while for 30 degrees still a high magnitude of 135 N is allowed.

The fatigue life for the screw is improved as much as the fixture became a component which is the most endangered to fatigue crack. Therefore, let us now concentrate on the influence of vertical load on its fatigue life. In the implant loaded with only horizontal load of 30 N there are two points with the highest effective stress amplitude exceeding 60 MPa (an effect of unloading or compression due to bending). They are located on the opposite sides of the fixture–abutment interface (Fig. 6). In accordance with the Goodman linear theory used in this study, the point with higher value of mean stresses is characterized by

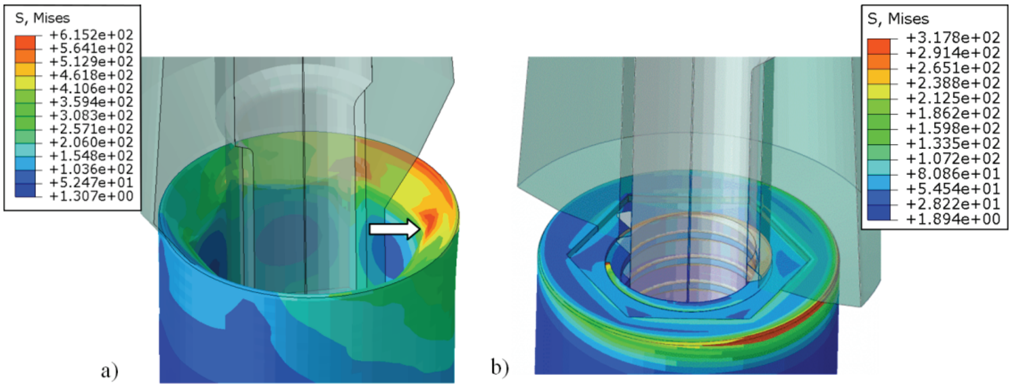


Fig. 6. Mises stresses in a fixture for horizontal load of 30 N in the initial (a) and optimized (b) design. Red areas indicate stress concentration

a lower number of fatigue cycles to failure (unloaded side). When vertical load is applied maximal Mises stresses on the compressed side increase significantly from 318 MPa (30 N/0 N) to 449 MPa (30 N/400 N) (Table 1). As a consequence, effective stress amplitude which is a combination of compression and bending effects reaches almost 100 MPa and moves the point of expected fatigue crack initiation to an opposite side of the interface. However, the compressive mean stress at this point is increased from 31 MPa to 313 MPa, which limits the increase of effective amplitude at zero mean stress. Thus, it changes only by 56%, while effective stress amplitude changes by 120%. Analyzing the vertical load influence without horizontal load, similarly to the screw, there is almost a linear relation between effective stress and vertical load magnitude. The influence of vertical load diminishes along with horizontal load increase (Fig. 3d). For the value of 60 N and 90 N the change of fatigue life is negligible. Summarizing the results for the fixture, the effect of vertical load introduction is analogous to that in the screw. Fatigue life is decreased if vertical load is present. For the highest value of vertical load of 400 N the number of cycles to fatigue fracture still exceeds 1.5e9.

Considering solely horizontal load in the initial design, the fixture seemed to be less endangered to fatigue fracture than the screw. However, analyzing the component for a wider spectrum of loads, the "safe" zone is smaller and meaningful region of functional load range overlaps "unsafe" zone. There are two mechanisms responsible for that. Firstly, the transverse load causes abutment rotation which opposes wedging in a hexagonal mortise. As a result, the stresses increase in an interior corner indicated by an arrow in Fig. 6. Secondly, the vertical load pushes the abutment into the fixture and due to conical contact surface it leads to the same effect. Therefore, although the vertical load stabilizes the abutment, the summarized effect of both components acting simultaneously is a reduction of fatigue life. The maximal magnitude of vertical load acting solely can reach almost 400 N without risk of fatigue fracture while the variation of load direction for implant axis reduces the allowed value to 170.4 N, 110.5 N and only 44,65 N for 5, 10 and 30 degrees, respectively.

Among the three components, the biggest improvement can be seen in the case of the fixture. Almost complete elimination of hexagonal slot along with horizontal contact surface between the fixture and the abutment causes that the fatigue fracture is not a problem anymore. The "safe" zone for this component covers the whole range of analyzed loads. What is more, the number of cycles to failure is higher tan 1e10 for all points within functional load range.

In the case of the last component, the abutment, fatigue life and Mises stresses in the optimized design increase when vertical load is applied for horizontal load up to 60 N. Higher value of the vertical load stabilizes the abutment. In the initial design an abutment is wedged into a hexagonal slot, which is the main reason for stress concentrations. Almost complete elimination of the slot in optimized design results in meaningful reduction of both maximal Mises stresses and fatigue fracture risk. The Mises stresses are below the limit of 282 MPa while the lowest number of cycles to fatigue failure is always higher than 6.1e13 (the worst case is 90 N/0 N of horizontal/vertical load) and 1e15 considering only functional load range. Thus, the influence of vertical load as well as a higher value of horizontal load can be neglected from the practical point of view.

5. Conclusions

The obtained results prove that coronial-apical load component should be taken into consideration in the analyzed two component dental implant when fatigue life is considered. In some range of analyzed loads compressive load reduces screw preload in the screw increasing stress amplitude during a load cycle. Furthermore, in a dental fixture the vertical load pushes in the abutment into the fixture and, due to conical contact surface between these components, makes stress concentration in hexagonal slot higher. Both phenomena, in turn, decrease fatigue life. From practical point of view, however, the reduction is not significant in the analyzed optimal design. It can be "safely" used within the assumed range of functional load and, with small exception for screw component, within the whole analyzed physiologically justified load range. Yet, it has to be noted that the optimal design is a concept only – the technological issues of production and requirements of medical procedures were not taken into consideration. Therefore, the results can slightly vary when necessary modifications are implemented.

A wider perspective for different combinations of load component magnitudes also allows us to make some interesting observations. Firstly, although in the initial design undergoing pure bucco-lingual load a screw is the weakest element, the "safe" area for the implant fixture is significantly smaller. The cyclic compression of the wedged abutment exceeds an influence of screw preload fluctuation to fatigue life decrease. Thus, hexagonal slot although beneficial to loosening resistance can cause fatigue life decrease. Secondly, as long as occlusal load direction agrees with implant axis, there is no risk of fatigue fracture. Thirdly, an abutment "safe" zone overlaps entirely the functional load range for both the initial and optimized design and, therefore, no fatigue fracture risk is observed for this component within functional load range. Finally, the optimization improves fatigue life for all components within the whole analyzed range of loads moving the "safe" limit away from functional load range.

Acknowledgements

This article was financially supported within the project "Engineer of the Future. Improving the didactic potential of the Poznań University of Technology" – POKL.04.03.00-00-259/12, implemented within the Human Capital Operational Programme, co-financed by the European Union within the European Social Fund.

References

[1] Adell R., Eriksson B., Lekholm U., Brånemark P.I., Jempt T., *Long-term follow-up study of osseointegrated implants in the treatment of totally edentulous jaws*, International Journal of Oral and Maxillofacial Implants, 1990, 5(4), 347–359.

[2] Alkon I., Sertgo A., Ekici B., *Influence of occlusal forces on stress distribution in preloaded dental implant screws*, Journal of Prosthetic Dentistry, 2002, 15, 38–42.

[3] Bozkaya D., Muftu S., *Mechanics of the taper integrated screwed-in (TIS) abutments used in dental implants*, Journal of Biomechanics, 2005, 38, 87–97.

[4] Ekici B., *Numerical analysis of a dental implant system in three-dimension*, Advanced in Engineering Software, 2002, 33, 109–113.

[5] Flanagan D., Llies H., McCullough P., McQuoid S., *Measurement of the fatigue life of mini dental implants: a pilot study*, The Journal of Oral Implantology, 2008, 34(1), 7–11.

[6] Freitas A., Bonfante E., Martins L., Silva N., Marotta L., Coelho P., *Reliability and failure modes of anterior single-unit implant-supported restorations*, Clinical Oral Implants Research, 2012, 23(9), 1005–1011.

[7] Freitas A., Almeida E., Bonfante E., Silva N., Coelho P., *Reliability and failure modes of internal conical dental implant connections*, Clinical Oral Implants Research, 2013, 24(2), 197–202.

[8] Genna F., *On the effects of cyclic transversal forces on osseointegrated dental implants: experimental and finite element shakedown analyses*, Computer Methods in Biomechanics and Biomedical Engineering, 2003, 6(2), 141–152.

[9] Genna F., *Shakedown, self-stresses, and unilateral contact in dental implant problem*, European Journal of Mechanics A/Solids, 2004, 23, 485–498.

[10] Garcia D., Zysset P.K., Charlebois M., Curnier A., *Internal vs. external connections for abutments/reconstructions: a systematic review*, Clinical Oral Implants Research, 2012, 6, 202–16.

[11] Ilies H.T., Flanagan D., McQuoid S., *Determining the Fatigue Life of Dental Implants*, Journal of Medical Devices, 2008, 2(1).

[12] Jung R.E., Pjetursson B.E., Glauser R., Zembic A., Zwahlen M., Lang N.P., *A systematic review of the 5-year survival and complication rates of implant-supported single crowns*, Clinical Oral Implants Research, 2008, 19(2), 119–130.

[13] Kayabasi O., Yuzbasioglu E., Erzincanli F., *Static, dynamic and fatigue behaviors of dental implant using finite element method*, Advanced in Engineering Software, 2006, 37, 649–658.

[14] Mericske-Stern R., Ventez E., Fahrlander F., Burgin W., *In vivo force measurements on maxillary implants supporting a fixed prosthesis or an over denture: A pilot study*, Journal of Prosthetic Dentistry, 2000, 84, 535–547.

[15] Merz B.R., Hunenbart S., Belser U.C., *Mechanics of the implant-abutment connection: an 8-degree taper compared to a butt joint connection*, The International Journal of Oral and Maxillofacial Implants, 2000, 15(4), 519–526.

[16] Morneburg T.R., Proschel P.A., *Measurement of misticatory forces and implant load: a methodological clinical study*, International Journal of Prosthodontics, 2002, 15, 20–27.

[17] Morneburg T.R., Proschel P.A., *In vivo forces on implants influenced by occlusal scheme and food consistency*, International Journal of Prosthodontics, 2003, 16, 481–486.

[18] Pessoa A., Starita A., *Influence of Implant Connection Type on the Biomechanical Environment of Immediately Placed Implants – CT-Based Nonlinear, Three-Dimensional Finite Element Analysis*, Clinical Implant Dentistry and Related Research, 2010, 12(3), 219–234.

[19] Piermatti J., Yousef H., Luke A., Mahevich R., Weiner S., *An in vitro analysis of implant screw torque loss with external hex and internal connection implant systems*, Implant Dentistry, 15, 427–435.

[20] Ribeiro C., Maia M., Scherrer S., Cardoso A., Wiskott H., *Resistance of three implant-abutment interfaces to fatigue testing*, Journal of Applied Oral Science, 2011, 19(4), 413–420.

[21] Steinebrunner L., Wolfart S., Ludwig K., Kern M., *Implant–abutment interface design affects fatigue and fracture strength of implants*, Clinical Oral Implants Research, 2008, 19, 1276–1284.

[22] Szajek K., *Optimization of a two-component implantology system using genetic algorithm*, Poznań: Publishing House of Poznań University of Technology, 2012.

[23] Wierszycki M., Kąkol W., Łodygowski T., *The screw loosening and fatigue analyses of three dimensional dental implant model*, Proc. of the Abaqus Users' Conference, Boston 2006.

[24] Wierszycki M., *Numeryczna analiza wtyrzymałościowa wszczepów uzębienia oraz segmentu kręgosłupa ludzkiego*, PhD thesis, Poznań 2007.

[25] Zheng L., Yang J., Hu X., Luo J., *Three dimensional finite element analysis of a novel osteointegrated dental implant designed to reduce stress peak of cortical bone*, Acta of Bioengineering and Biomechanics, 2014, 16(3), 21–28.