https://doi.org/10.31436/iiumej.v24i2.2695

A COMPARATIVE FINITE ELEMENT ANALYSIS OF REGULAR AND TOPOLOGICALLY OPTIMISED DENTAL IMPLANTS FOR MECHANICAL AND FATIGUE RESPONSES EVALUATION

MUHAMMAD IKMAN ISHAK[1*], RUSLIZAM DAUD[1], BAKRI BAKAR[1] AND SITI NOOR FAZLIAH MOHD NOOR[2]

[1]Faculty of Mechanical Engineering & Technology, Universiti Malaysia Perlis, Kampus Alam UniMAP, Pauh Putra, 02600 Arau, Perlis, Malaysia
[2]Advanced Medical and Dental Institute, Universiti Sains Malaysia, Bertam, Jln. Tun Hamdan Sheikh Tahir, 13200 Kepala Batas, Pulau Pinang, Malaysia

[*]Corresponding author: ikman@unimap.edu.my

(Received: 19 December 2022; Accepted: 29 May 2023; Published on-line: 4 July 2023)

ABSTRACT: Topology optimisation is a prominent method to improve the performance of any systems by optimising geometrical factors to save materials without compromising the system functionality. Currently, there is limited published data discussing the topologically optimised dental implants that makes the matter still unclear. This study aimed to evaluate the mechanical and fatigue behaviours of regular and topologically optimised dental implant designs using 3-D FEA. Geometrical models were developed in accordance with ISO 14801 using SolidWorks 2020 before being analysed in ANSYS 18.1. The new implant design was created by topology optimisation analysis. The material properties of all parts were assumed to be isotropic, linearly elastic, and homogenous. Nine different compressive load values ranging from 100 to 500 N were applied on the loading structure as separated cases. The vertical and bottom surfaces of the holder were fully constrained. The results showed that the topologically optimised implant recorded about 12.3% lower implant stress than the regular implant. Both implant designs revealed a comparable displacement result with a percentage difference of only 2.3%. The optimised design was also found to produce longer fatigue life and approximately 12.3% higher safety factor compared to the regular design. The increase in the compressive load value has increased the stress and deformation, whilst decreased the fatigue life and safety factor in both designs. Although it was estimated that the volume of the new implant could be reduced to about 24% of the traditional one, the implant functionality may still be retained or even be improved.

ABSTRAK: Pengoptimuman topologi adalah kaedah utama bagi meningkatkan prestasi mana-mana sistem dengan mengoptimumkan faktor geometri bagi menjimatkan bahan tanpa menjejaskan fungsi utama sistem. Dewasa ini, terdapat kurang data diterbitkan berbincang mengenai implan gigi yang dioptimumkan secara topologi yang menjadikan perkara ini masih tidak jelas. Kajian ini bertujuan bagi menilai perlakuan mekanikal dan kelesuan bagi reka bentuk implant gigi biasa dan yang dioptimumkan secara topologi menggunakan 3-D FEA. Model geometri telah dibangunkan mengikut ISO 14801 menggunakan SolidWorks 2020 sebelum dianalisis dalam ANSYS 18.1. Reka bentuk implan baharu telah dibuat melalui analisis pengoptimuman topologi. Sifat pada semua bahagian bahan diandaikan sebagai isotropik, keanjalan linear, dan homogen. Sembilan nilai beban mampatan berbeza antara 100 hingga 500 N telah dikenakan pada struktur pembebanan sebagai kes berasingan. Permukaan menegak dan bawah pemegang dikekang sepenuhnya. Keputusan menunjukkan bahawa implan yang dioptimumkan secara topologi

merekodkan tegasan implan 12.3% lebih rendah daripada implan biasa. Kedua-dua reka bentuk implan menunjukkan hasil anjakan yang setanding dengan perbezaan peratusan hanyalah 2.3%. Reka bentuk yang dioptimumkan juga didapati menghasilkan hayat kelesuan yang lebih lama dan kira-kira 12.3% faktor keselamatan yang lebih tinggi berbanding reka bentuk biasa. Peningkatan dalam nilai beban mampatan telah meningkatkan tegasan dan perubahan bentuk, sementara mengurangkan hayat kelesuan dan faktor keselamatan dalam kedua-dua reka bentuk. Walaupun dianggarkan bahawa isipadu implan baru boleh dikurangkan kira-kira 24% daripada implan tradisional, fungsi implan masih boleh dikekalkan atau dipertingkatkan.

KEYWORDS: *deformation; dental implant; fatigue; finite element analysis; topology optimisation*

1. INTRODUCTION

Endosseous dental implants are a popular treatment choice for partially or completely edentulous patients to restore dental performance and improve dental aesthetic appearance [1]. This artificial surgical part is always in high demand and reported to possess a satisfactory success and survival rate [2]. Titanium is the most common material for dental implants due to its biocompatible properties allowing the implant to remain stable and function successfully. The root form implant is one of the types of the endosseous dental implant that occupies the vertical bone region mimicking the root structure of a natural tooth [3]. Among the typical configurations of the root form implant are cylindrical- and screw-type where both are available in solid or hollow form. The hollow-cylindrical implant is introduced to increase the total contact surfaces between the implant and the bone compared to the solid-cylindrical implant. The solid-cylindrical design is exposed to destructive shear force and dependent on surface treatment for improving retention to the bone. The solid-screw implants, alternatively, possess macroscopic retaining features along the body for primary bone anchorage. Apart from reducing the impact of shear load, the implant retentive elements may also decrease the potential of overloading at the bone-implant interface. The solid-screw implants offer an improved functional contact surface with the adjacent bone and easier surgical placement compared to hollow-cylindrical or hollow-screw designs.

Biomechanical compatibility of a dental implant is influenced by many factors such as the applied occlusal loading, implant geometry, implant dimensions, implant material stiffness, and bone quality and quantity [4]. The connection of the implant with the attached living tissues is indicated by a physiological phenomenon that is described as osseointegration. In the early stage of implantation, it is normally observed that peri-implant bone resorption occurs due to less mechanical stimulation. As the high stress value and concentration can exist in the vicinity of the implant, particularly at the implant neck region, this could also predict the occurrence of bone loss [5]. It has been suggested that in low-quality bone, the implant with large diameter, long, and straight body wall is highly preferable. Besides, the application of a platform switching concept in dental implants, where the diameter of the abutment is much smaller than the implant platform, produces lower tensile and compressive bone stresses than the platform-matched implant [6]. The platform-switched implants could preserve alveolar bone level by shifting the stresses from the area of compact bone to the area of trabecular bone. In terms of the applied load and bone quality, these two factors are challenging to alter for an optimum implant stress response at the bone-implant interface. Many attempts have been made to optimise the design of dental implants including by resembling the shape of a natural tooth. The optimised implant design with a tapered and wider neck exhibited considerably lower peak stresses relative to the traditional implant design [7].

Topology optimisation provides an optimised material allocation in a given design space based on a set of constraints and loadings [1,8,9]. In other words, it is a method for shape optimisation to establish material outline. An ideal design of structure demands an even stress dissipation within the body up to tolerable limit. There is a possibility to remove material at the minimally stressed regions such that the final design could achieve similar function with a reduced mass. For instance, a topologically optimised design of spinal cage managed to reduce the volume of existing design about 36% and increase the space for bone augmentation while maintaining afforded spinal stability [10]. Regarding dental implants, apparent high stress concentration is commonly observed at the regions interfacing with the compact bone with minimal stress accumulation at the implant apex. The mathematical algorithm in topology optimisation technique is in reference to objective function, design variables, and constraints [11]. Through finite element analysis (FEA), the optimisation method iteratively analyses the design performance to remove the redundant materials without affecting the structural stability and function. The FEA has been proven as a useful and widely accepted method to solve complex mathematical problems regarding stability and failure analyses in many fields such as structure, biomedicine, electronics, fluid, and heat. The use of FEA in implant dentistry began in 1973, and has since been increasingly used to predict responses that are challenging to obtain in experimental and clinical works. To date, limited published data could be found on the topologically optimised solid dental implant that makes this matter still unclear and inconclusive. In a study by Chang et al. [8], it was reported that the volume of the original implant design could be decreased about 17.9%. The study analysed the implant embedded in the maxillary first molar region and applied with a static occlusal load. A more recent investigation by Gupta et al. [1] demonstrated a volume reduction of 32 – 45% could be achieved with the implant still able to retain its function. The variation in the length and diameter of the implant and bone quality were studied under the application of constant static load. Dental implants are highly exposed to fatigue failure due to repetitive occlusal loading. Many existing computational studies have only evaluated the performance of traditional implants and analysed the optimum topology of an implant under static loading, without emphasising on fatigue characteristics. Critical grasp of fatigue conditions is important to comprehend the force transfer within a dental implant system.

The main objective of the present study was to examine and compare the mechanical and fatigue responses between a regular and topologically optimised dental implant designs under different vertical compressive load values using three-dimensional (3-D) FEA. Four types of result criteria, namely maximum equivalent von Mises stress, maximum total deformation, minimum fatigue life, and minimum safety factor were interpreted from the models. The null hypothesis of this study was that the traditional and new implant designs demonstrate insignificant difference in the response data investigated. The novelty of our study was to shed light on the determination of redundant material distribution in a threaded dental implant and generation of clearer quantitative mechanical and fatigue response data for regular and topologically optimised implant designs. This may give valuable insights for clinicians and/or implant manufacturers in the development of new implant macrogeometry design with reduced mass and acceptable strength. It is also expected that this study could provide an improved comprehension of the force transfer in different implant designs under the evaluation of fatigue conditions.

2. MATERIALS AND METHODS

2.1 Geometrical Model

The geometry of a commercial dental implant system (dual-fit (DFI)) from Alpha-Bio Tec, Petach Tikva, was used as a reference to build a 3-D model of a solid threaded dental implant. The length and diameter of the implant body are 11.5 and 3.75 mm, respectively. Besides, a 3.5-mm high straight abutment, and a 2.2-mm wide and 8.0-mm long abutment screw were also modelled. The abutment screw is used to rigidly attach the abutment to the implant body at the implant platform. In this study, the implant platform was designed with an internal hexagonal connection as stated in the implant manufacturer's catalogue. Apart from the implant components, the 3-D geometrical modelling also includes a holder (40.9 mm (L) x 39.8 mm (H) x 23.0 mm (W)), a hemispherical cap (6.5 mm (L) x 5.1 mm (W)), and a loading structure (12.0 mm (L) x 11.0 mm (W)). As the study covers the fatigue prediction, the 3-D model setup was prepared based on the fatigue testing standard ISO 14801 using a computer-aided design (CAD) software, SolidWorks 2020 (SolidWorks Corp., Concord, Massachusetts, USA). Related in-built geometrical shape creation tools provided in the software such as extrude, mirror, revolve, sweep, and/or loft were employed to develop the models.

All the models were assembled and placed under the testing condition described in ISO 14801. Figure 1(a) exhibits the exploded view of the final geometrical analysis model. The implant body was virtually embedded into a 4.5-mm diameter cylindrical hole created on the holder. Boolean subtraction option was adopted to place the implant body into the holder. A bone loss of 3.0 mm measured from the implant platform to the holder surface (Fig. 1(b)) was simulated, and the abutment and its screw were attached to the implant body. The hemispherical cap was then placed onto the abutment for loading. The central axis of the cap, abutment, abutment screw, and implant body were ensured to be aligned. The central point of the cap was set to be at 11.0 mm from the bone (holder) level to represent a moment arm. The assembled model was then transferred into ANSYS 18.1 software (ANSYS Inc., Houston, TX, USA) to generate the model mesh and pre-processing settings of the FEA.

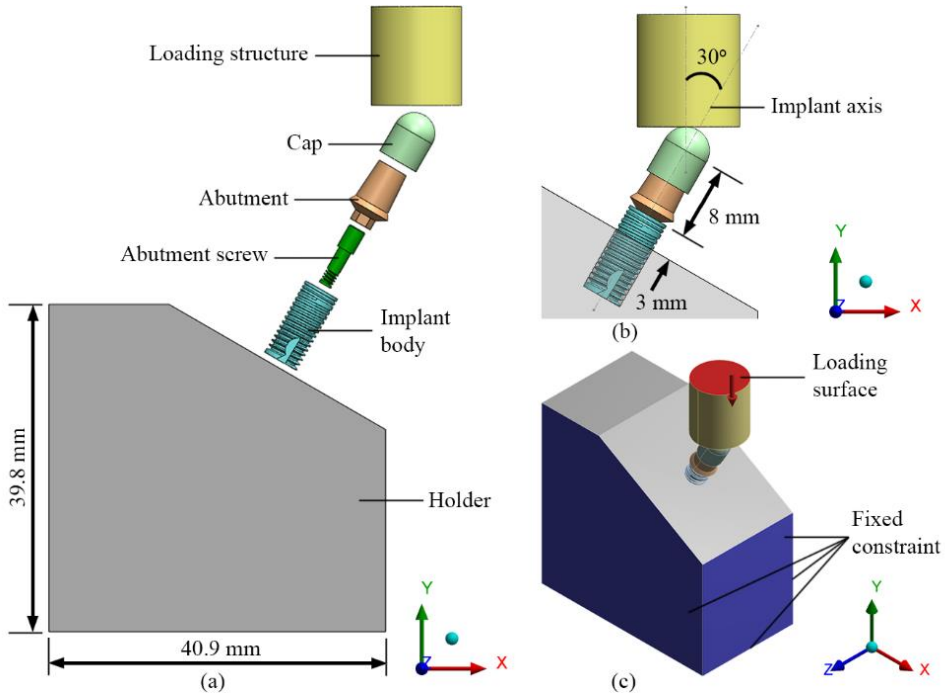

Fig. 1: (a) Exploded view of the model. (b) Dimensions for the implant placement. (c) Loading and support locations of the model.

2.2 Material Properties

Each component in the model was considered to be isotropic, homogenous, and linearly elastic. The abutment, abutment screw, and implant body were made of titanium alloy (Ti-6Al-4V) with a yield strength of 847 MPa [12]. Additionally, the hemispherical cap and loading structure were assigned with steel alloy. According to ISO 14801 specifications, the implant body must be fixed in a rigid clamping holder having the elastic modulus of at least 3.0 GPa. Therefore, in this study, we have chosen aluminium alloy as the material of the holder. A summary of the material properties for the model setup is shown in Table 1.

Table 1: Material properties for the model setup

Material	Part	Young's Modulus, *E* (GPa)	Poisson's Ratio, *v*	References
Ti-6Al-4V	Abutment, abutment screw & implant body	113.8	0.342	Yalçın et al. [13]
Aluminium alloy	Holder	71	0.33	Bayata et al. [14]
Steel alloy	Cap & loading structure	200	0.31	Yao et al. [15]

2.3 Interface Conditions

All the interfaces were modelled as perfectly bonded using contact and target elements in ANSYS software. This type of contact modelling was also implemented in many previous related studies [16]. The bonded contact type indicates no penetration or loosening among the interfaces of the implant body-holder, abutment screw-implant body, abutment-abutment screw, abutment-implant body, cap-abutment, and cap-loading structure.

2.4 Loading and Boundary Condition

As recommended by ISO 14801, to simulate the experimental test condition, a vertical load was applied on the flat surface of the loading structure model. This load configuration basically represents an oblique load at an inclination of 30° with respect to the implant longitudinal axis [17]. The applied compressive load was varied from 100 to 500 N with an increment of 50 N that signify the range of normal bite forces [18]. The location of the loads was at a distance of 11.0 mm from the holder inclined surface.

Fatigue analysis was performed on the implant complex to predict the mechanical responses and corresponding fatigue behaviours of the titanium alloy components. The analysis was executed by a repetitive simulated masticatory loading sets (as stated earlier) with alternating load value using fatigue algorithm based on Goodman fatigue theory in the elasticity mode using ANSYS software. The fatigue lives and potential failure regions were predicted for infinite fatigue life criteria.

For the model boundary condition, all the nodes on the vertical and bottom surfaces of the holder were constrained in all degrees of freedom. This denotes that the nodal displacement is set to zero at the stated surfaces. Figure 1(c) depicts the loading and boundary condition used in the analysis.

2.5 Topology Optimisation

In clinical practices, most dental implant designs used have a circular cross-sectional area. This kind of design possesses low resistance towards shear or torsional forces especially when the abutment screw is tightened or even when the implant is free-standing. The inclusion of antirotational features such as vent or hole in the apical part of the implant body is expected to increase inner space for bone ingrowth while not losing too much

stiffness. Topology optimisation tool in ANSYS software was employed to construct a new dental implant design that targeted to increase internal contact surfaces of the implant body. It is important to note that the strength of the implant body must be retained with the decreasing volume.

The main goal of topology optimisation is to reduce the structure compliance energy. Reducing the compliance is similar to increasing the global stiffness of the structure. In other words, the standard formulation of the method is minimising the structure compliance and at the same time satisfying a limitation on the structure volume. The topology optimisation problem is described as follows (Eq. (1) to Eq. (6)):

$$\text{objective function} = \text{minimise}\left(U_c\right) \tag{1}$$

$$\text{limitation} = 0 < \eta_i < 1 \left(i = 1,2,3,\ldots,n\right) \tag{2}$$

$$V \le V_0 - V^* \tag{3}$$

$$V = \sum \eta_i V_i \tag{4}$$

$$E_i = E\left(\eta_i\right) \tag{5}$$

$$\left\{\sigma_i\right\} = \left[E_i\right]\left\{\varepsilon_i\right\} \tag{6}$$

where U_c = energy of structure compliance; η_i = internal pseudodensities that are assigned to each finite element (i) in the topology problem; V = computed volume; V_0 = original volume; V^* = amount of material to be removed; V_i = volume of element i; E_i = elasticity tensor for each element; E is the elasticity tensor; i = stress vector of element i; ε_i = strain vector of element i.

The variables of density η range from 0 to 1 in which a value close to 1 indicates that the material should be kept, while a value close to 0 indicates material should be removed. In the optimisation analysis, the volume of the entire regular implant body (design domain) was set to be decreased by about 50% with 10 iterations as the pre-set response constraints. The convergence accuracy was defined as 0.1%. By applying 100-N vertical static force normal to the implant platform and fixedly constrained the outer surfaces along the implant body, we managed to obtain a reasonable distribution of material for a new implant design as illustrated in Fig. 2. It was observed that only the internal portion of the apical implant body volume was removed. A comparison of mechanical and fatigue behaviours between regular and topologically optimised implants were made and evaluated.

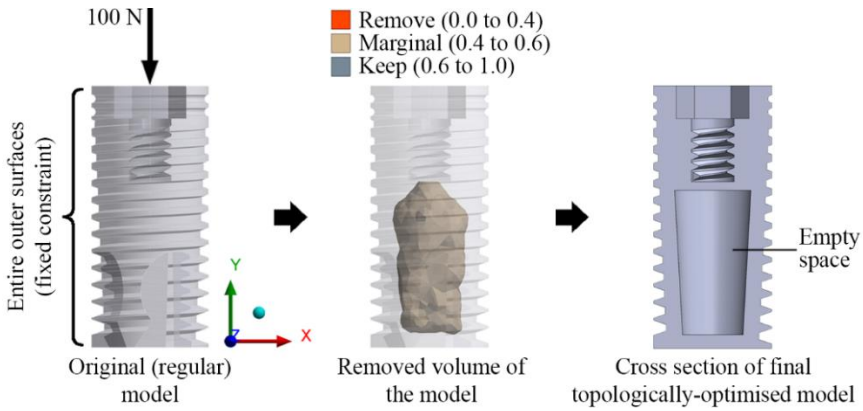

Fig. 2: Main stages of topology optimisation analysis.

2.6 Model Meshing

Since the geometrical shapes of the implant parts are irregular, complex, and small in dimensions, ten-node quadratic tetrahedral elements (SOLID187) were applied in this study to assure the continuity of the force and displacement transfers on the nodes, as considered in some earlier studies [19,20]. Prior to final model discretisation, it is worth mentioning that a mesh convergence analysis was performed so that the analysis results are independent of mesh condition. A smaller mesh size provides a better estimate for the exact solution. However, a high number of nodes and elements may increase computing time. Thus, an agreement must be achieved between the ideal mesh size and the demand for a reliable solution. In this mesh independence test, the details of material properties, contact modelling, and boundary conditions are as mentioned in the previous sections with the 100-N applied load. The maximum equivalent von Mises stress within the abutment-implant complex was recorded for variety mesh sizes. A total of nine relative characteristic mesh sizes was considered which are 2.2 mm (Tet A: ~34,000 elements), 2.0 mm (Tet B: ~43,000 elements), 1.8 mm (Tet C: ~67,000 elements), 1.4 (Tet D: ~114,000 elements), 1.2 mm (Tet E: ~117,000 elements), 1.0 mm (Tet F: ~300,000 elements), 0.95 mm (Tet G: ~349,000 elements), 0.8 mm (Tet H: ~581,000 elements), and 0.7 mm (Tet I: ~865,000 elements). Upon refining the mesh, the acceptability of the result is determined based on the variation of maximum stress value among the mesh sizes, which should be less than 5% [21, 22]. In general, there was a significant difference found on the stress values generated among the mesh sizes. After four refinements, the result seemed to yield at the mesh size of 1.2 mm (Tet-E) with the relative deviation of 1.2%. The total number of nodes and elements are approximately 250,000 and 176,500, respectively. The illustration of mesh sensitivity plot and mesh distribution in the coarse (Tet-A) and refined (Tet-E) models are shown in Fig. 3.

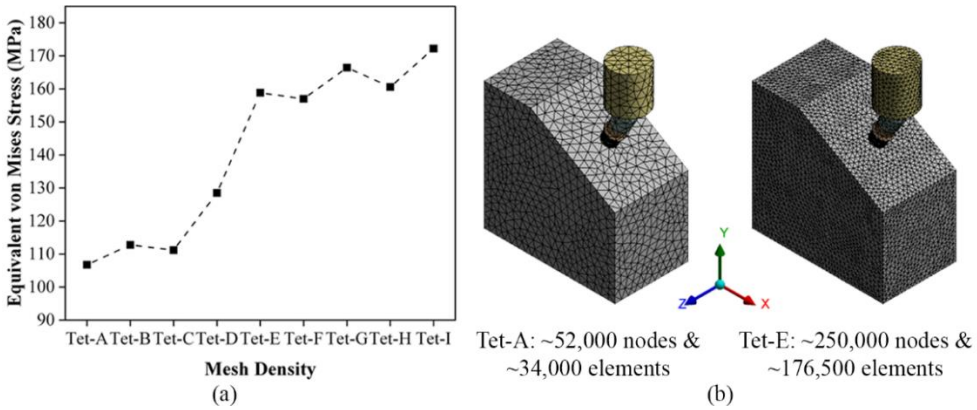

Fig. 3: (a) Mesh sensitivity plot for different element numbers. (b) Mesh distribution in the coarse (Tet-A) and refined (Tet-E) models.

2.7 Model Verification

A comparison was made between our proposed model with those published in earlier studies that considered similar analysis type, modelling design, and analysis software. The pre-processing settings used in the selected previous studies were replicated as closely as possible. It was found that our implant stress result was comparable with that of previous studies as shown in Table 2. A little difference was recorded that could possibly be due to different detailed geometrical features of the models and several assumptions made.

https://doi.org/10.31436/iiumej.v24i2.2695

Table 2: Comparison of implant equivalent von Mises stress results between literature and our models

Past Studies	Literature Results	Proposed Model Results
Wang et al. [16]	368.4 MPa	326.19 MPa
Cheng et al. [23]	625.21 MPa	462.02 MPa
Bayata et al. [14]	368.68 MPa	392.02 MPa

3. RESULTS

3.1 Topology Optimisation Analysis

The new implant design was successfully developed from the topology optimisation analysis. As depicted in Fig. 2, only the apical third part of the implant body was affected, and the corresponding volume was removed. The reduction of volume was about 24% considering the change from 123.13 mm^3 (traditional implant) to 93.63 mm^3 (new implant). Meanwhile, the finite element model of the new complex implant was reconstructed and consisted of 249,500 nodes and 176,000 elements.

3.2 Mechanical Behaviour of the Regular and Topologically Optimised Implant Designs

The magnitude of the maximum equivalent von Mises stress and maximum total deformation of the implant-abutment complex for all loading values was extracted in the post analyses to predict the risk of implant failure. The colour contour outline of the results was also provided to clearly scrutinise the mechanical stimuli distribution.

3.2.1 Maximum Equivalent von Mises Stress Results

In comparison, the regular or standard implant design exhibited a greater maximum equivalent von Mises stress value within the implant-abutment assembly than the topologically optimised one, irrespective of loading levels (Fig. 4(a)). The increase in the load value led to the linear increase in the output stress for both designs with the peak values of 794.23 MPa and 707.12 MPa recorded in the regular and optimised designs, respectively, under 500-N load. Meanwhile, the lowest stress was generated by 100-N load with the value of 158.85 MPa (regular design) and 141.42 MPa (optimised design). It became evident that the regular design recorded about 12.3% higher implant stress as compared to the new design in all loading values.

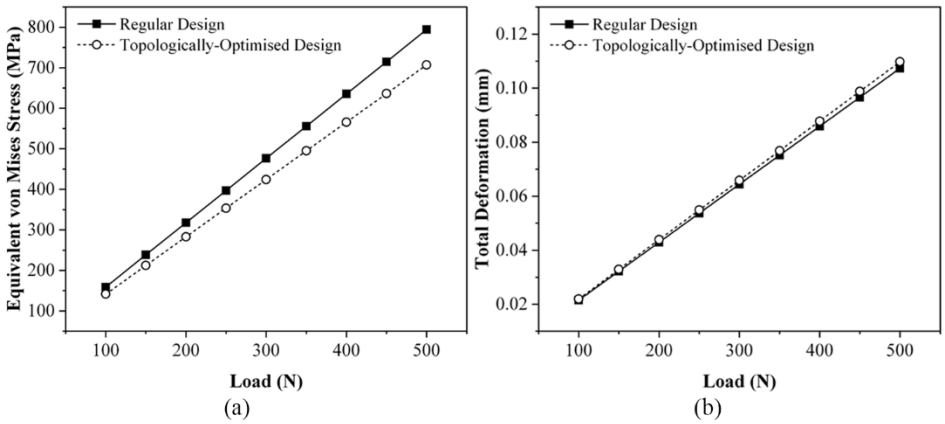

Fig. 4: (a) Maximum equivalent von Mises stress and (b) total deformation of the implant-abutment assembly for both designs under all loading conditions.

https://doi.org/10.31436/iiumej.v24i2.2695

Figure 5 depicts the maximum stress concentration was found in the abutment specifically at the interface region of the implant platform. Besides, a high stress amplification region was also observed around the upper threads of the implant body and near the junction of the holder regardless of implant design. The distribution of stress within the assembly was comparable between the regular and new designs for all loading values.

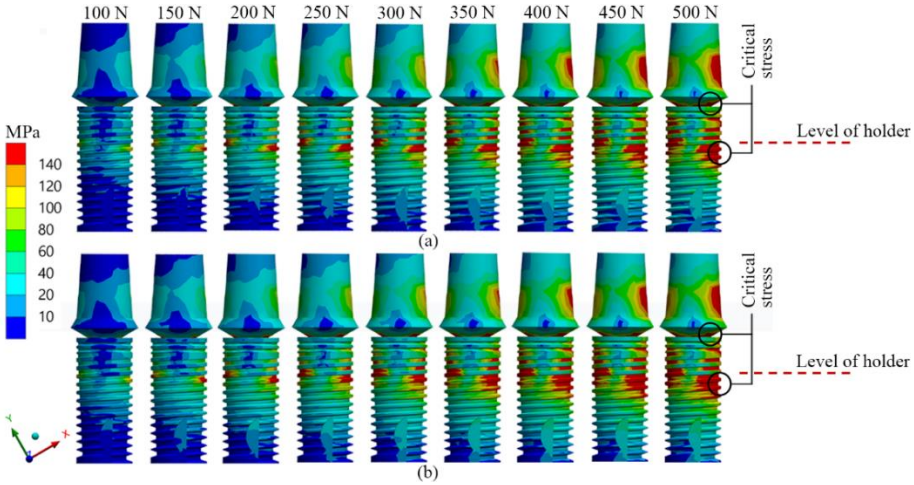

Fig. 5: Dissemination of equivalent von Mises stress in the implant-abutment complex for (a) regular and (b) topologically optimised designs.

3.2.2 Maximum Total Deformation Results

The displacement of the implant-abutment assembly for the regular form was relatively lower than that for the topologically optimised design in all loading values (Fig. 4(b)). Similar to the stress results, it was shown that the total deformation was proportionally increased with the increase in the loading value for both implant configurations. The load of 500 N resulted in the maximum deformation of the implant-abutment assembly with the value of 0.1073 and 0.1098 mm for the regular and new implant designs, respectively. The minimum deformations were 0.0215 mm (regular design) and 0.022 mm (optimised design) generated under 100-N load. The displacement of the new model was only increased by about 2.3% compared to that of the regular model regardless of loading values.

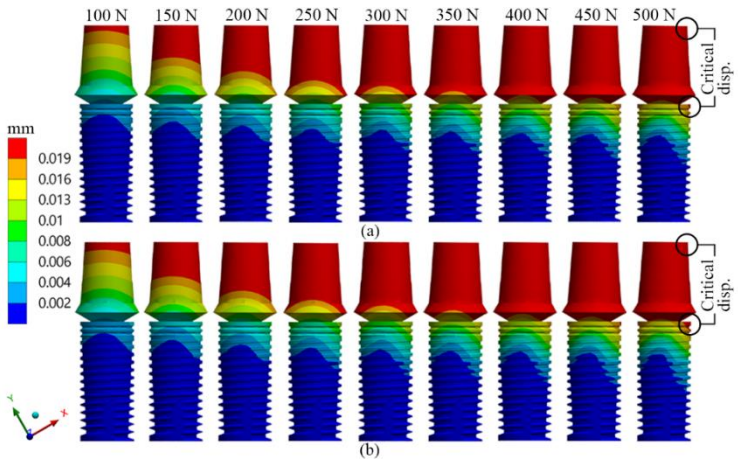

Fig. 6: Distribution of total deformation of the implant-abutment complex for (a) regular and (b) topologically optimised designs.

3.3 Fatigue Behaviour of the Regular and Topologically Optimised Implant Designs

The minimum value of fatigue lives and safety factor of both implant designs under different loading magnitudes were presented. The outcomes were also interpreted in the form of spectrum colouring scale for a clearer understanding.

3.3.1 Minimum Fatigue Life Results

Figure 7(a) shows a comparison of minimum fatigue life (cycles) between the regular and topologically optimised designs. Our results demonstrated that fatigue failure was only predicted for the applied loading values ranging from 350 – 500 N by considering the fatigue life limit of 5 x 10^6 cycles based on existing experimental fatigue testing. These loads were equivalent to the resulting stresses of 555.96 – 794.23 MPa and 494.99 – 707.12 MPa for the regular and new implant designs, respectively. It was observed that the regular design showed a shorter estimated life with the lowest cycle number of 0 as compared to the optimised design that yielded at 2.69 x 10^5 cycles under 500-N load. There was no difference in the number of cycles (1 x 10^7 cycles) at the lower loading values (100 – 300 N) among both designs.

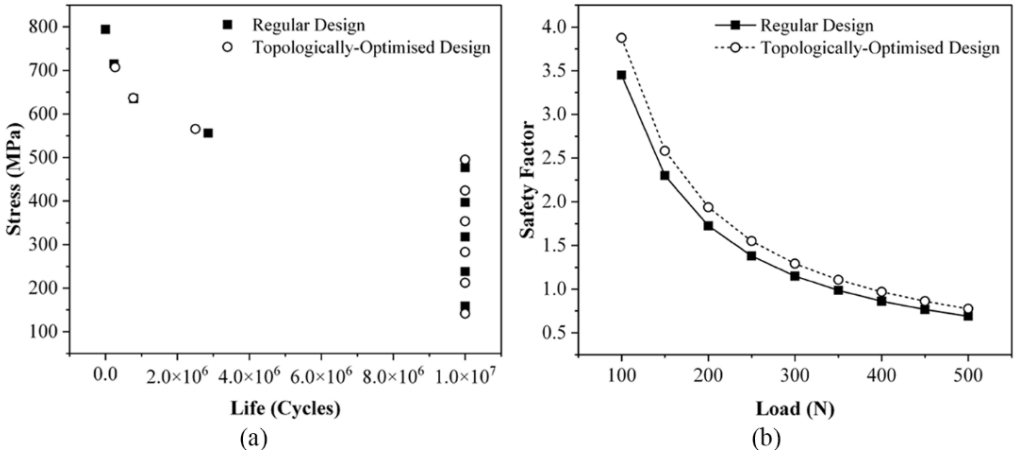

Fig. 7: (a) Minimum fatigue life and (b) safety factor of the implant-abutment assembly for both implant designs under all loading conditions.

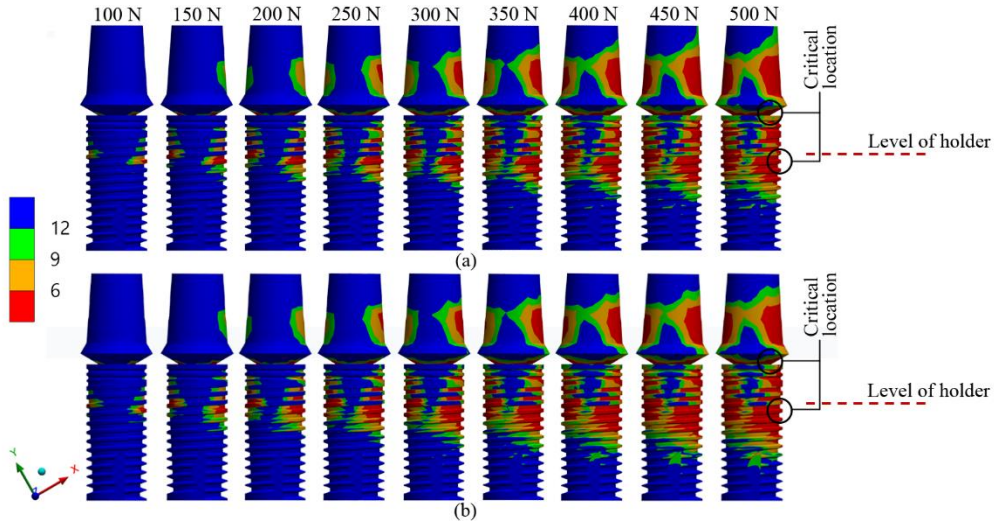

Fig. 8: Distribution of safety factor of the implant-abutment complex for (a) regular and (b) topologically optimised designs under all loading conditions.

3.3.2 Minimum Safety Factor Results

The safety factor values for fatigue life were computed based on Goodman fatigue theory in accordance with infinite fatigue life criteria. The plot of minimum safety factor in the implant-abutment complex versus the applied loads is depicted in Fig. 7(b). The safety factors for the new design were greater than those for the regular design irrespective of loading values. The maximum safety factors were 3.45 and 3.87 for the regular and optimised designs, respectively, under the load of 100 N. It is noted that the safety factors were below 1 when the models were subjected to the load from 350 N for the regular design, and from 400 N for the optimised design. This finding was consistent with the fatigue life results where the load lower than 350 N was predicted to be safe against fatigue. It was observed that critical regions that could attribute implant to failure were the abutment-implant connection and the middle threads of the implant body as illustrated in Fig. 8. Safe regions were more dominant at the apical portion of implant body and top part of the abutment.

4. DISCUSSION

Dental implants are used to transmit the occlusal force to the neighbouring bones. Therefore, the main functional design objective can be associated with the distribution of external loads by improving the function of implant-supported prosthesis. The integration of fundamental scientific knowledge related to geometrical features and force with engineering solutions may accomplish the targeted clinical goals. Presently, there is a high number of studies investigating the perseverance of bone-implant attachment in order to obtain optimal mechanical stimuli transfer. However, low emphasis is placed, and limited information available, on the structural optimisation design of dental implants, particularly for fatigue prediction. Our topology optimisation analysis showed that approximately 24% of the whole implant body volume was redundant material. Only the internal volume of the apical part was removed while the other regions were kept intact. The suggested regions to be removed also covered the vicinity underneath the threaded hole for the abutment screw attachment. However, this was not completely removed so as to preserve the appropriate volume in sustaining the screw integrity and to minimise the potential of high stress concentration in that region. One study performed by Chang et al. [8] reported that the decreased volume of the traditional implant as a result of topology optimisation was lower compared with the one of our findings where the reduction percentage was only about 17.9%. In contrast, the finding from Gupta et al. [1] predicted a far greater volume reduction of 32 – 45% for a topology study that also considered the effect of the implant macrogeometry and bone quality. The differences could possibly be due to the different geometrical shape of the models and pre-processing settings of the analysis employed. Nevertheless, the consistency was found for the location of the removed material which is the apical part of the implant body. To examine the mechanical and fatigue behaviours of the regular and topologically optimised implant designs, linear static FEA was performed under a variation of occlusal loading values to analyse the stress and displacement of the implant-abutment complex.

The level of stress in the newly modified implant was about 12.3% lower than the one in the traditional implant. However, the pattern of stress dissemination over the regions in the assembly seemed to be identical in both designs. Our findings disagreed with the results of Chang et al. [8] where the optimised design (128.1 MPa) generated 13% higher implant stress value compared to the original one (113.3 MPa) under the 204-N oblique load. In contrast, the findings of Gupta et al. [1] were consistent with ours by which a lower implant

stress was recorded by the modified design (33.21 MPa) than the traditional design (54.26 MPa) at the imposed volume reduction of 50%. A possible explanation of our results is that the reduction of implant material tends to minimise the stress shielding effect. Therefore, the mechanical stress transfer from the high stiff implant to the surrounding region having lower stiffness is increased. As a result, the implant-abutment embodiment for the optimised design sustained less stress. In terms of the region of stress distribution, apparent stress amplification occurred at the connecting part of the abutment, and at the implant neck near the junction of the holder. This is parallel with the stress colour contour plots in past studies [1, 8]. One can conclude that the removal of volume from the end of the implant body up to its mid region can be made. A study by Shi et al. reported that alternative implant design with a larger and tapered crestal part radius was preferred compared to the commercially available implant due to a lower peak stress generated [7]. Besides, the topologically optimised implant design could increase the area for osseointegration ingrowth by providing more spaces in the apical region [24]. To relate our results with more realistic studies that concern biological situations, it is commonly found that high stress intensity is located at the interface with the compact bone or peri-implant bone region. A bone stress higher than the strength of the cortical bone, 170 MPa, means a prediction of failure. In implant dentistry, peri-implant bone loss is one of the main manifestations regarding osseo-insufficiency which could lead to detrimental implications such as patient complaints, aesthetic compromise, soft and hard tissue deformation, and implant removal [25]. A greater decrease in the bone level was noticed in the first year of implantation of about 1.0 mm; however, it was reduced to 0.2 mm on average in the following years. The loosening and fracture of the implant due to the resorption of supporting bone could cause eventual implant failure. As such, the effect of bone loss should be minimised by achieving optimum mechanical stress transfer at the bone-implant interface. In this study, titanium alloy Ti-6Al-4V is the main material used for the implant components in both implant designs. We found that the peak implant stress regardless of implant designs under all loading values (141. 42 – 794.23 MPa) was lower than the yield strength of Ti-6Al-4V (847 MPa) [12]. This would predict the success of the implants or low tendency of failure.

In implant dentistry, rigid fixation of an implant is desired as it is the criterion of successful treatment. Rigid fixation is defined as the attainment of implant stability without clinical mobility or the displacement evaluated with horizontal or vertical forces, similar to analysing natural teeth. A healthy natural tooth moves about 56 – 73 μm [26], while a firm implant displaced less than 75 μm [27] with no clinical movement. The implant is exposed to a higher risk of failure than a natural tooth if its motion is higher than 0.5 mm horizontally. Our results revealed that a higher displacement occurred in the abutment rather than in the implant body in both designs which could be due to less resistance towards direct loading imposed from the cap. The optimised design showed a slightly greater abutment deformation relative to the regular one that possibly caused by the reduced retention on the apical portion of the implant body. The loss of volume in that region attributed the implant body to instability, thus increasing the tendency of dislocation. Concerning the displacement of the implant body itself, it was evident that the peak value of the optimised and regular designs was 22.2 and 20.7 μm, respectively, under the 500-N applied load. Compared to the motion of a healthy implant, the values were relatively 70.4% (optimised) and 72.4% (regular) lower than 75 μm. The optimised design still offered acceptable rigid fixation to maintain implant stability although it recorded about 2.3% greater deformation than the regular design.

For the fatigue behaviour assessment, one cyclic loading consisted of one 30° oblique loading [17]. The traditional and topologically optimised models survived up to 1 x 10^7

https://doi.org/10.31436/iiumej.v24i2.2695

cycles under different loading limits which are 300 N and 350 N, respectively. Further to that, under the highest loading value of 500 N, a total failure was found for the regular implant, while the new implant yielded at 2.69 x 10^5 cycles. According to ISO 14801, a dental implant can sustain a life of at least 5 x 10^6 without demonstrating any damage [14]. Our results therefore corresponded well with the standard. The optimised implant design was expected to have higher resistance to fatigue failure in their life as compared to the regular design.

The safety factor of the implant-abutment assembly was above 1 for the loading magnitudes up to 300 N and 400 N in the regular and optimised models, respectively. It is determined by the relation between mean stress, σ_m and alternating stress, σ_a according to the modified Goodman theory as,

$$\sigma_m = \frac{\sigma_{\max} + \sigma_{\min}}{2} \tag{7}$$

$$\sigma_a = \frac{\sigma_{\max} - \sigma_{\min}}{2} \tag{8}$$

$$\frac{\sigma_a}{S_e} + \frac{\sigma_m}{S_u} = \frac{1}{N_f} \tag{9}$$

where S_e = endurance limit, S_u = ultimate tensile strength of the material, and N_f = the safety factor for fatigue life in a loading cycle. The safety factor values were decreased in the regular implant compared to the optimised implant due to the increased stress level in the regular implant. The most critical value of 0.69 was recorded in the regular implant under the applied load of 500 N. Thus, considering the safety factor in the mechanical aspect, the topologically optimised implant is more favourable for dental implant application.

Albeit that robust outcomes of the study were found, the quantitative result data were still predictive and could be relying on several limitations of the analyses. In fact, the implant was placed in the non-living material under static loading, while the real applied one is embedded in the complex living tissues without a definite pattern, and this argument may somewhat affect the results. Besides, this study only analysed the single restoration type, thus the findings were attributed to this kind of treatment. Several important aspects can be considered in future topology optimisation studies such as assessing different material stiffnesses and dimensions of the implant body, employing more complex geometrical models, considering more realistic dynamic occlusal loading, and simulating implant extraction from the bone socket. Although the outcomes of the present study could not directly be inferred to actual clinical situation, they managed to reveal a difference in the mechanical and fatigue behaviours through simulation analysis. *In vitro* and *in vivo* studies are needed in order to validate the prognosis of implant perseverance even at low levels. The null hypothesis of the study was rejected as the traditional and topologically optimised implant designs demonstrated a significant difference in the responses obtained.

5. CONCLUSION

The new implant designed by topology optimisation analysis has reduced the volume of the traditional implant by about 24%. However, this topologically optimised design was still able to maintain the implant fixation in sustaining the loadings.

- It was shown that the newly modified implant generated nearly 12.3% lower implant stress than the regular one.

- In terms of displacement, insignificant difference was found among both designs where the percentage deviation was merely 2.3%.
- The optimised implant design also promoted a longer fatigue life and an improved safety factor by approximately 12.3%.
- The manufacturing cost of the new implant might be high using conventional machining processes. However, it could alternatively be manufactured using additive manufacturing methods, which is expected to offer much lower cost.

ACKNOWLEDGMENT

The authors would like to acknowledge the support from Fundamental Research Grant Scheme (FRGS) under a grant number of FRGS/1/2020/TK0/UNIMAP/03/2 from the Ministry of Higher Education Malaysia. The authors reported no conflicts of interest related to this study.

REFERENCES

[1] Gupta Y, Iyer R, Dommeti VK, Nutu E, Rana M, Merdji A, Biswas JK, Roy S. (2020) Design of dental implant using design of experiment and topology optimization: a finite element analysis study. Proc Inst Mech Eng H: J Eng Med., 235(2): 157-166. https://doi.org/10.1177/0954411920967146

[2] Agustin-Panadero R, Leon-Martinez R, Labaig-Rueda C, Faus-Lopez J, Sola-Ruiz MF. (2019) Influence of implant-prosthetic connection on peri-implant bone loss: a prospective clinical trial with 2-year follow-up. Int J Oral Maxillofac Implants., 34(4): 963-968. https://doi.org/10.11607/jomi.7168

[3] Samieirad S, Mianbandi V, Shiezadeh F, Hosseini-Abrishami M, Tohidi E. (2019) Tapered versus cylindrical implant: which shape inflicts less pain after dental implant surgery? A clinical trial. J Oral Maxillofac Surg., 77(7): 1381-1388. https://doi.org/10.1016/j.joms.2019.02.027

[4] Bidez MW, Misch CE. (2015) Chapter 5 - Clinical biomechanics in implant dentistry. In Dental Implant Prosthetics. 2nd Edition. St. Louis, Mosby; pp 95-106.

[5] Gil J, Sandino C, Cerrolaza M, Pérez R, Herrero-Climent M, Rios-Carrasco B, Rios-Santos JV, Brizuela A. (2022) Influence of bone-level dental implants placement and of cortical thickness on osseointegration: In silico and in vivo analyses. J Clin Med., 11(4): 1027 https://doi.org/10.3390/jcm11041027

[6] Chang CL, Chen CS, Hsu ML. (2010) Biomechanical effect of platform switching in implant dentistry: a three-dimensional finite element analysis. Int J Oral Maxillofac Implants., 25(2): 295-304.

[7] Shi L, Li H, Fok AS, Ucer C, Devlin H, Horner K. (2007) Shape optimization of dental implants. Int J Oral Maxillofac Implants., 22(6): 911-20.

[8] Chang C-L, Chen C-S, Huang C-H, Hsu M-L. (2012) Finite element analysis of the dental implant using a topology optimization method. Med Eng Phys., 34(7): 999-1008. https://doi.org/10.1016/j.medengphy.2012.06.004

[9] Seitz KF, Grabe J, Köhne T. (2018) A three-dimensional topology optimization model for tooth-root morphology. Comput Methods Biomech Biomed Engin., 21(2): 177-185. https://doi.org/10.1080/10255842.2018.1431778

[10] Zhong Z-C, Wei S-H, Wang J-P, Feng C-K, Chen C-S, Yu C-H. (2006) Finite element analysis of the lumbar spine with a new cage using a topology optimization method. Med Eng Phys., 28(1): 90-98. https://doi.org/10.1016/j.medengphy.2005.03.007

[11] Zhang K, Cheng G, Xu L. (2019) Topology optimization considering overhang constraint in additive manufacturing. Comput Struct., 212: 86-100. https://doi.org/10.1016/j.compstruc.2018.10.011

[12] Lee H, Jo M, Noh G. (2021) Biomechanical effects of dental implant diameter, connection type, and bone density on microgap formation and fatigue failure: A finite element analysis. Comput Methods Programs Biomed., 200: 105863. https://doi.org/10.1016/j.cmpb.2020.105863
[13] Yalçın M, Kaya B, Laçin N, Arı E. (2019) Three-dimensional finite element analysis of the effect of endosteal implants with different macro designs on stress distribution in different bone qualities. Int J Oral Maxillofac Implants., 34(3): e43–e50. https://doi.org/10.11607/jomi.7058
[14] Bayata F, Yildiz C. (2020) The effects of design parameters on mechanical failure of Ti-6Al-4V implants using finite element analysis. Eng Fail Anal., 110: 104445. https://doi.org/10.1016/j.engfailanal.2020.104445
[15] Yao K-T, Chen C-S, Cheng C-K, Fang H-W, Huang C-H, Kao H-C, Hsu M-L. (2018) Optimization of the conical angle design in conical implant–abutment connections: a pilot study based on the finite element method. J Oral Implantol., 44(1): 26-35. https://doi.org/10.1563/aaid-joi-D-17-00149
[16] Wang K, Geng J, Jones D, Xu W. (2016) Comparison of the fracture resistance of dental implants with different abutment taper angles. Mater Sci Eng C., 63, 164-171. https://doi.org/10.1016/j.msec.2016.02.015
[17] Zhang X, Mao J, Zhou Y, Ji F, Chen X. (2020) Study on statics and fatigue analysis of dental implants in the descending process of alveolar bone level. Proc Inst Mech Eng H., 0954411920926080. https://doi.org/10.1177/0954411920926080
[18] Levartovsky S, Peleg G, Matalon S, Tsesis I, Rosen E. (2022) Maximal bite force measured via digital bite force transducer in subjects with or without dental implants - A pilot study. Appl Sci., 12(3): https://doi.org/10.3390/app12031544
[19] Amruta G, Rajeev S, Gaurang M, Simran M, Mishal Ds, Kunal M. (2022) Comparative evaluation of stress on the peri-implant bone of different densities in mandible restored by various diameter of implants - a finite element analysis. J Electron Sci., 12(11): 161-177.
[20] Satheesh Kumar P, Satheesh KK, John J, Patil G, Patel R. (2013) Force transfer and stress distribution in an implant-supported overdenture retained with a hader bar attachment: a finite element analysis. ISRN Dent., 2013, 369147. https://doi.org/10.1155/2013/369147
[21] Niroomand MR, Arabbeiki M. (2020) Implant stability in different implantation stages: analysis of various interface conditions. Inform Med Unlocked, 19: 100317. https://doi.org/10.1016/j.imu.2020.100317
[22] Jeng M-D, Lin Y-S, Lin C-L. (2020) Biomechanical evaluation of the effects of implant neck wall thickness and abutment screw size: a 3D nonlinear finite element analysis. Appl Sci., 10(10), 3471: https://doi.org/10.3390/app10103471
[23] Cheng Y-C, Jiang C-P, Lin D-H. (2019) Finite element based optimization design for a one-piece zirconia ceramic dental implant under dynamic loading and fatigue life validation. Struct Multidiscipl Optim., 59(3): 835-849. https://doi.org/10.1007/s00158-018-2104-2
[24] Lawrence AA, Singh N, Davis R, Sahil Ansari M, Tewari YV, Francis V. (2022) Chapter 13 - Topological design optimisation of dental implant. In Optimization of Industrial Systems. 1st Edition. Edited by Panchal D, Tyagi M, Sachdeva A, Pamucar D. Scrivener Publishing LLC.
[25] Naveau A, Shinmyouzu K, Moore C, Avivi-Arber L, Jokerst J, Koka S. (2019) Etiology and measurement of peri-implant crestal bone loss (CBL). J Clin Med., 8(2): 166. https://doi.org/10.3390/jcm8020166
[26] Rudd KD, O'Leary TJ, Stumpf AJ Jr. (1964) Horizontal tooth mobility in carefully screened subjects. Tech Doc Rep US Air Force Syst Command Electron Syst Div, 94: 1-6.
[27] Sekine H, Komiyama Y, Potta H, Yoshida K. (1986) Mobility characteristics and tactile sensitivity of osseointegrated fixture-supporting systems. In Tissue Integration in Oral Maxillofacial Reconstruction. 1st Edition. Edited by Van Steeberghe D. Amsterdam, Excerpta Medica; pp. 326–332.