G Model
JORMAS-794; No. of Pages 12
ARTICLE IN PRESS
J Stomatol Oral Maxillofac Surg xxx (2020) xxx–xxx
Available online at
ScienceDirect
www.sciencedirect.com
Elsevier Masson France
EM|consulte
www.em-consulte.com/en

Original article

Design methodology for dental implant using approximate solution techniques

O. Kayabasi

Department of Biomedical Engineering, Duzce University, Konuralp Yerleskesi Merkez/Duzce, 81620 Turkey

ARTICLE INFO

Article history:
Received 5 October 2019
Accepted 14 January 2020

Keywords:
Dental Implant
Probabilistic Approach
Finite Element Analysis
Structural Optimization

ABSTRACT

With the developing technology, dental implants have been widely used in recent years. These implants are surgically implanted into a jaw bone to support missing teeth. Implants are usually made of titanium and are biocompatible. The design and analysis of the dental implant is based on expert knowledge, experience and ability to work seamlessly on the patient. Due to the difficulties in performing dental implant tests in vivo, the geometric shape design of the dental implant must be performed before it is applied to a patient and mathematical models have been developed to perform structural analysis. In this study, a design strategy for dental implant design was proposed. In this proposed strategy, finite element analysis, numerical optimization method and probabilistic design approach Monte Carlo simulation are integrated to work together automatically.

1. Introduction

Dental implants have been successfully applied to the patient in recent years to treat decaying teeth. These implants are surgically implanted in a jawbone to support missing teeth or teeth. Implants are usually made of titanium and are bio-compatible. [1,2]. The practice of dental implants is so complex that optimum design needs to be established between engineers and surgeons by determining design parameters. Therefore, engineers and surgeons must work together to create a durable and reliable dental implant. Because of the natural nature of the bone, it strongly influences the design of the dental implant. The success, design, and analysis of the dental implant depends on expert knowledge, experience and ability to work seamlessly on the patient. Because of the difficulties in performing dental implant tests in vivo, the strength calculations of the dental implant should be made prior to application to a patient. Approximate solution techniques have been developed to perform structural analysis before administration to the patient to reduce the cost of in vivo tests. Therefore, the tooth implants can be designed in the computer environment and structural analysis can be performed before they are attached to the patient.

One of the mathematical models developed is the Finite Element Method (FEM). This method is considered to be the most advanced simulation technique used in biomechanical applications in recent years with the development of computer technologies and the development of mathematical models. This method provides many advantages over simulating the complexity of clinical conditions and other methods. Stress distributions in the jaw bone can be used to estimate the stress distributions and displacements occurring in the dental implant. This method allows a large number of complex scenarios to be studied in a computerized environment before the dental implant is applied to the patient. This will minimize the time it takes to design the dental implant and prevent permanent damage from improper design and application of the dental implant.

Dental implants have been studied in literature on many different topics. In these topics, we can list the implant material, structural analysis of the implant, and biological effects of the implant Our study explains all these criteria, since our success depends on the integration of the implant into the jawbone, the strength of the interface between the bone and the implant, and the strength of the life of the entire patient.

Finite element analysis (FEA) has been widely used to assess the distribution of stress on the implant and around the bone [3,4]. It was used first in implant dentistry in 1973 by Tesk and Widera. It has become useful in the dental implant-jaw interface, in the circumference of the jawbone, in the estimation of the stress distribution in the jawbone and implant [5]. Many of the previous finite element analysis studies have independently examined the effect of parameters of the dental implant [6]. Implant designs included the diameter of the implant, the pitch of the thread, and the length of the implant. These criteria have been accepted as key factors in implant design [7–9]. The failure or success of dental

E-mail address: oguzkayabasi@duzce.edu.tr

https://doi.org/10.1016/j.jormas.2020.01.003
2468-7855/

Please cite this article in press as: Kayabasi O. Design methodology for dental implant using approximate solution techniques. J Stomatol Oral Maxillofac Surg (2020), https://doi.org/10.1016/j.jormas.2020.01.003

implant is defined as the conduction of jawbone and bones around the implants in the implant interface [10,11]. Screw design plays an important role in dental implant design and is one of the most effective factors [12]. Over the years, many experimental and clinical studies have shown that the first tooth in contact with the jawbone is the maximum stress zone, and thus the initial micro-fracture has come to this region. Finally immature osseointegration occurs in this region. There are many commercial dental titanium implants that are commercially available but differ in their screw designs. Understand the role of stress distribution in the cortical surrounding screw thread angles and spongy bones were analyzed using 3D Finite Element Method for four different implant designs with varying tooth angles [13]. In order to evaluate the effect of bone type on bone density and cortical bone thickness, stresses induced by two implants under pressure and curved loads were applied a numerical simulation technique based on the finite element method. Two types of implants (M-12 and Astra Tech) were introduced in a model matrix whose geometry was extracted from a true CBCT [14]. Research on mini dental implant induced stress and tension in the peri-implant bone were investigated when implant loading was performed [15]. Stress analyzes were carried out using finite element analysis which was divided into 3 groups according to the thread shape of the implants registered by American Patent Institute and having 15 licenses. The results of the dental implant designed by the researchers were compared [16].

In this study, a design strategy was proposed to reduce the cost of in vivo tests and to design a reliable dental implant. In this proposed strategy, Finite element analysis, numerical optimization algorithm and Monte Carlo Simulation are applied to work together automatically. Using this solution approach, the geometric design of the dental implant is formulated with a numerical optimization algorithm. In calculating the optimization problem obtained from the results of the analysis of the finite element analysis of the implant, objective and restrictive functions are changed approximately before the optimization problem is solved. At the end of the optimization problem, the optimal design of the implant has emerged. Using the results of this study, for optimally optimized dental implant designs with minimal stress, the probability of failure was investigated using several simple performance functions that dynamically define the fatigue theory of the bone-dental implant interface. The reliability of the dental implant was estimated using a written program based on the ANSYS finite element analysis program. Optimal dental implant performance compared to original design performance. After obtaining the optimum dental implant shape, the dental implant was manufactured according to the geometric shape obtained to prove the design methodology we developed. The experimental measurement was performed according to the dental implant test procedures.

Treatment with dental implants can be a very complicated stage in terms of planning, conducting and managing the following problems. Although this treatment has a high success rate, these are unknown and are managed with best avoidance without correcting after applying to the patient. The aim of this chapter is very important, starting with the first consultation of this approach, including the techniques to be applied in case of difficulties.

2. Methods

2.1. Probabilistic methods and Monte Carlo simulation

After obtaining the best geometric shape, the probability analysis of the dental implant was calculated as follows:

$$Z(X) = Z(X1, X2, X3, .., Xn) \tag{1}$$

Z (X) is a random variable describing the stress and strain response at the nodes and elements in the finite element model, Xi (i = 1, n) refers to random variables that define input variables in the model. In this study, Z variable is obtained from the results of finite element analysis. Monte Carlo sampling techniques were used for the probability response. In fact, Monte Carlo is a technique for predetermining what future outcomes an event will produce with certain possibilities. Once the input/parameters of the specified event are loaded into the system, a model is established and the data is entered at random times to obtain certain possibilities for the outcome. In this way, we will see the results it brings to us without putting the system into effect at a lower cost.

Failure in engineering design is typically evaluated in conventional deterministic analyzes by comparing the calculated response of a structure (e.g. stress) with the calculated strength of the material, usually with an associated safety factor. In Monte Carlo simulation, which is a probabilistic analysis solution, the probability of error of the system can be expressed by formulating a performance function, which is the function of random variables by comparing the probabilistic response of the structure with the probabilistic strength of the material [17].

The failure risk at this stage was calculated according to the performance function of the species.

$$g(X) = R(X) - S(X) \tag{2}$$

Here *g(X)* represents the failure event of the system. When *R(X)*-is a random function that describes the strength or resistance of the component, *S(X)* is the response of the structure (e.g. stress) and is also a random variable. X is the vector of random variables. The probability of failure is defined as shown in equation 3.

$$p_f = P(g(X) \leq 0) \tag{3}$$

Performance functions are formulated for dental implant and bone implant and bone interface. In this study, eight performance functions have been selected for performance functions.

$$g_1 = R_{UCS} - S\sigma_{\text{dental implant}}(X) \text{ Compressive failure} \tag{4}$$

$$g_2 = R_\tau - ST_{\text{rdental implant}}(X) \text{ Shear failure} \tag{5}$$

$$g_3 = R_{FL} - S\sigma_{\text{dental implant}}(X) \text{ Fatigue failure} \tag{6}$$

$$g_4 = R = -S\sigma = (X) \text{ Compressive failure} \tag{7}$$

$$g_5 = R_\tau - ST_{\text{rbone}}(X) \text{ Shear failure} \tag{8}$$

$$g_6 = R_{FL} - S\sigma_{\text{bone}}(X) \text{ Fatigue failure} \tag{9}$$

$$g_7 = R_T - S\sigma_{\text{dental implant bone interface}}(X)\text{Tensile failure} \tag{10}$$

$$g_8 = R_\tau - S\sigma_{\text{dental implant bone interface}}(X)\text{Shear failure} \tag{11}$$

Rτ, R_{UCS}, R_{FL} are shear strength, tensile strength and fatigue limit strength of bone and dental implant, respectively. S(X) is the measure of response obtained from the results of three-dimensional finite element analysis. The probability of failure and probability response were measured both from the results of the deterministically optimized tooth implant geometry and from the initial design. Random variable statistics are defined in the random variable definition window in the computer program written by us. The numerical model in the program allows the use of the finite element program ANSYS or a user-defined model. Using linear regression in the definition of the regression model allows the transfer of function coefficients or data sets to the program as standard linear or quadratic functions.

Monte Carlo is an experimental method that aims to produce random numbers. With this method, it is aimed to solve mathematical and physical problems. The method is based on probability theory. In this method, it is important to simulate and solve an experiment with statistical and mathematical techniques or a physical phenomenon, which must be solved by repeatedly generating random numbers. The main purpose of this method is to extract the properties of large elements by means of a randomly selected subset. For example, the expected value of any function f (x) in the range (a, b) is intended to be subtracted from the estimated value at the points of the randomly selected end number in this range.

2.2. Parametric modeling

If the goals and requirements for investigating the optimal dental implant geometry to meet our needs are expressed in terms of the definition of an optimization problem as follows, can be automated with a numerical optimization algorithm [18]. The parameterised model is shown in Fig. 1.

In this study, eight geometric parameters are defined as real variables as shown in Fig. 1. These parameters are screw head diameter P_1, conic surface opening angle of screw head P_2, screw head height P_3, screw diameter P_4, hexagonal slot height P_5, conic surface opening angle of hexagonal slot P_6, thread shapes of dental implant (right) P_7, thread of pitch P_8.

Find design parameters:

$$P_1, P_2, P_3, P_4, P_5, P_6, P_7, P_8 \quad (12)$$

to minimize design objective:

$$\text{Maximum Stress(von Misesstress)} \quad (13)$$

subjected to design constraints:

$$\sigma_{implant} \leq \sigma_{implant}^{Yield} \quad (14)$$

$$\sigma_{bone} \leq \sigma_{bone}^{Yield} \quad (15)$$

$$\tau_{implant-bone} \leq \tau_{implant-bone}^{failure} \quad (16)$$

$$N_{implant} \geq 1 \quad (17)$$

within the design space:

$$0.9 \leq P_1 \leq 1.8 \quad (18)$$

$$10 \leq P_2 \leq 85 \quad (19)$$

$$1.2 \leq P_3 \leq 7 \quad (20)$$

$$1 \leq P_4 \leq 1.7 \quad (21)$$

$$0.1 \leq P_5 \leq 3 \quad (22)$$

$$10 \leq P_6 \leq 90 \quad (23)$$

$$15 \leq P_7 \leq 90 \quad (24)$$

$$0.6 \leq P_8 \leq 1 \quad (25)$$

Equation 12 refers to design variables that define the geometry/shape of the dental implant. Equation 13 refers to the purpose of design to reduce the maximum stress on the entire dental implant system. Equations 14-17 specify design constraints on structural and fatigue strength limitations of dental implant components (i.e., dental implant-shaped, jawbone, dental implant-jawbone interface). Equations 18-25 express the limits of the minimum and maximum values of each design variable the limits of this design are defined by reference to experimental data and literature. The functions of the objectives and constraints in equations 14-17 are not explicitly (analytically) known before the solution of the optimization problem. For certain randomly chosen design parameter values corresponding to different dental implant shapes, the results of the finite element analysis were generated by applying the least squares method. Since the approaches of the objective and constraint functions are used, the optimization method is sometimes called the approximate optimization method, and the approaches are called response surface approach or response surface models. Linear and quadratic polynomial functions are often used to construct response surface approximations of purpose and constraint functions.

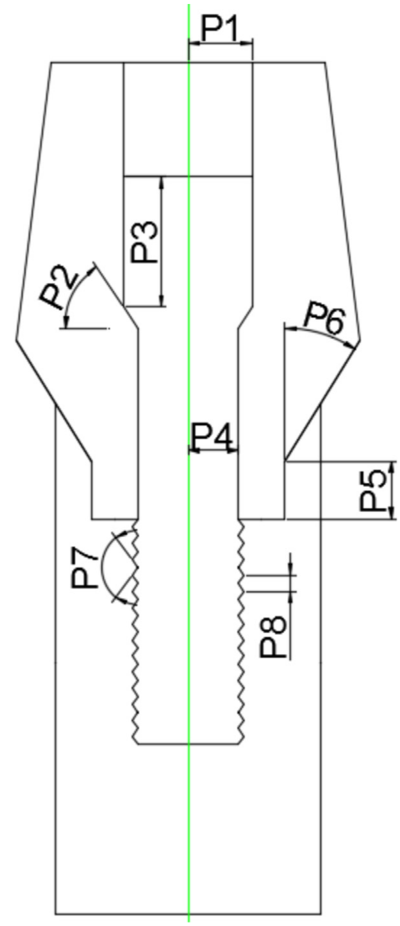


Fig. 1. Geometry of the dental implant and selected design parameters.

In this work, quadratic polynomial response surface functions $\tilde{y}(x)$, are used, as given by the following equation:

$$\tilde{y}(\mathbf{x}) = \underbrace{\underbrace{\underbrace{a_0 + \sum_{i=1}^{n} a_i x_i}_{linear} + \sum_{i=1}^{n} b_i x_i^2}_{quadratic} + \sum_{j=1}^{n-1} \sum_{i=j+1}^{n} c_{ji} x_j x_i}_{quadratic+crossterms} \quad (26)$$

where a, b, c are tuning coefficients to be determined and n is the number of design parameters.

The approximate solution optimization method is applied to the optimization of the dental implant through the ANSYS [19]

Design Optimization (DO) module. The flowchart of the approximate optimization method applied in this study is shown in Fig. 2.

2.3. Finite element method

The finite element model required for finite element analysis is based on the geometric model as shown in Fig. 3 and is divided into smaller and simpler elements. Mesh density was increased on these surfaces as the stresses would occur at the interface between the dental implant and jawbone during chewing. The FEM model consists of a total of 123,410 tetrahedron tetrahedron elements; 44.127 for implant, 13.534 for abutment, 4.987 for metal skeleton, 60.762 for bone. The bone implant, abutment, metal frame and tetrahedron elements correspond to the SOLID186 type elements in the ANSYS element library. FEM models are shown Fig. 3. It is expressed by the surface-to-surface contact algorithm feature of ANSYS to express the physical effects on the dental implant-jawbone interface. In this study, three different contact types were investigated to investigate the features of the jawbone and dental implant interface [20]. In the first study, the contact type was used completely bonded contact type second work, the friction coefficient was used as $\mu = 0.3$ and the contact type bonded in the third study was used with the friction coefficient $\mu = 0$.

In this study Ti-6Al-4 V was used for the implant fixture and abutment for the finite element model. As a material model, we use a linear isotropic material model to represent the behavior of the material. The probabilistic approach was applied to optimize the three-dimensional deterministic shape optimization of the dental implant with respect to the failure probability of the implant system. For this reason, random variable definitions of model input variables are determined from the literature and from actual experimental test data. Mechanical properties is used in this work is shown in Table 1.

To accurately assess the effect of bone behaviours on the implant, the outer and inner sides of the bone (cortical bone and spongy bone) are modelled using different material properties. The inner side of the bone is represented by the transversely isotropic material model ($Ex = Ey = 11.5$ GPa, $Ez = 17$ GPa, $Gxy = 3.6$ GPa, $Gxz = Gyz = 3.3$ GPa, $vxy = 0.51$, $vxz = vyz = 0.31$). The outer side of the bone was modeled as a linear isotropic material model with $E = 2.13$ and $\nu = 0.3$. Fatigue calculations of the implant are made for Ti-6Al-4 V alloyed material. In the fatigue calculations, the fatigue material model shown in Fig. 4 is used. Fig. 4, known as the S-N curves, shows the fatigue characteristics of the Ti-6Al-4 V alloy in terms of the tensile angle, which varies with the number of cycles.

The third step in the finite element model is to determine loading and boundary conditions. Random variable definitions of the model input variables were selected from the literature and actual experimental test data (Table 2). The FEM model was fixed on the surface of the mandibula as shown in Fig. 5.

In many studies, a load was applied in a direction as a boundary condition, but in this study, loading conditions for 5 seconds were applied as shown in Fig. 6 in order to reflect the reality of the physical environment that occurred during chewing in the mouth.

For this purpose, the target and contact surfaces between the individual parts of the model have been defined, without joining the nodes between the components. Contact elements were determined at a distance of 0.005 mm between implant teeth and bone, adjacent surfaces of the implant, and abutment and contacting surfaces. It provides the transfer of load and deformation between different components including contact analysis, assembly and friction coefficient of 0.3 [21]. The manufacturer recommended torque for implant placement, abutment connections are listed in Table 3. The preload was developed in the screw with a thermal load placed on the matching surfaces of the implant complex. The final element analysis of the dental implant was performed using the Intel core i5-4440 3.10 GHz processor in ANSYS on a PC. Each analysis takes about 20 hours of CPU time.

3. Results

In this paper, we have developed the design optimization methodology to evaluate the possibility of failure under different loading for different interface conditions with the dental implant, dental implant-bone interface. The performance function of the probabilistic analysis reveals the probabilities of finite failure in Table 4. All performance functions were examined and the most well-designed dental implant and the first designed dental implant results were compared. When the results are examined, the

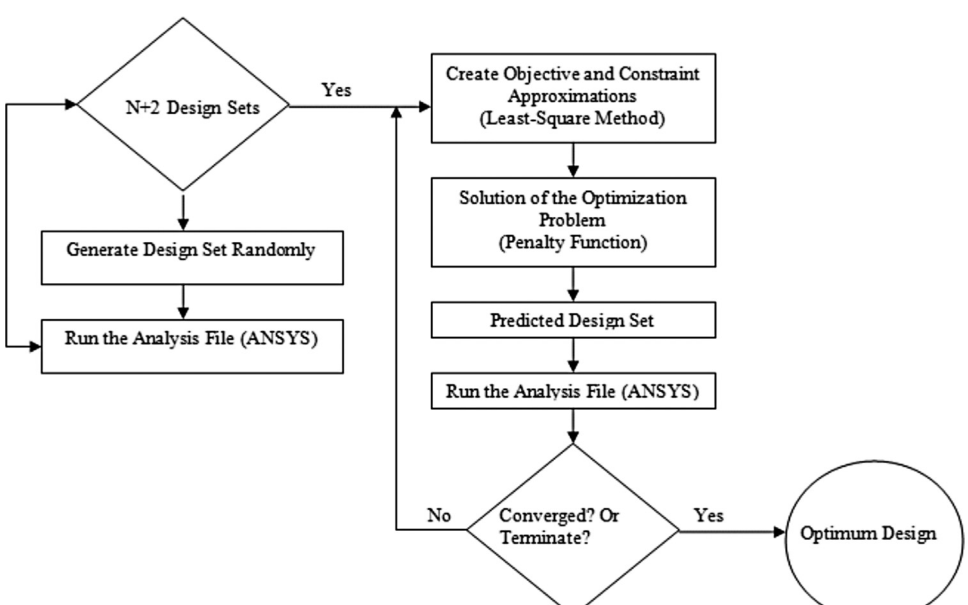


Fig. 2. Flow chart of the approximate optimization process with ANSYS DO module.

Fig. 3. FEM models of implant (a), abutment (b), metal framework (c) and occlusal material (d).

Table 2
Random variable for finite element model.

	Mean	Standard Deviations	Coefficient of variation (%)	Distribution Type
Mesio-distal direction (n)	23.45	3.74	15.93	Lognormal
Lingual direction (n)	17.16	3.15	18.44	Lognormal
Axial direction (n)	114.65	18.43	16.13	Lognormal

decrease in the probability of failure is seen as shown in Table 4. The first design results are shown in Fig. 7.

Fig. 8 shows the final shape of deterministic optimization results for the dental implant. Optimum shape, which is formed by minimizing the von Mises stress in dental implant, caused a decrease in the maximum value of von Mises stress in the form of dental implant compared to the first design. 30.89% reduction in maximal von Mises stress values in dental implant-bone interface was found with optimum dental implant shape. This optimum shape also reduced the maximum value of von Mises stress in the bone. Shape optimization resulted in a decrease in the mean value and standard deviation of each structural response in Table 5.

Table 1
Mechanical properties of materials.

	Mean	Standard deviations	Coefficient of variation (%)	Distribution type
Cortical bone Elastic Modulus (GPa)	20.5	2.45	11.5	Normal
Cancellous bone Elastic Modulus (GPa)	2.16	0.16	10.3	Normal
Ti-6Al-4V Elastic Modulus (GPa)	110	3.56	12.2	Lognormal
Dental Implant Shear Strength (MPa)	650	2.49	12.5	Normal
Dental Implant Compression Strength (MPa)	850	0.23	11.9	Normal
Dental Implant Fatigue Limit (MPa)	350	0.28	8.8	Lognormal
Bone Shear Strength (MPa)	22	2.41	10.84	Lognormal
Bone Compression Strength (MPa)	47	0.27	9.91	Lognormal
Bone Fatigue Limit (MPa)	18	0.23	9.28	Lognormal

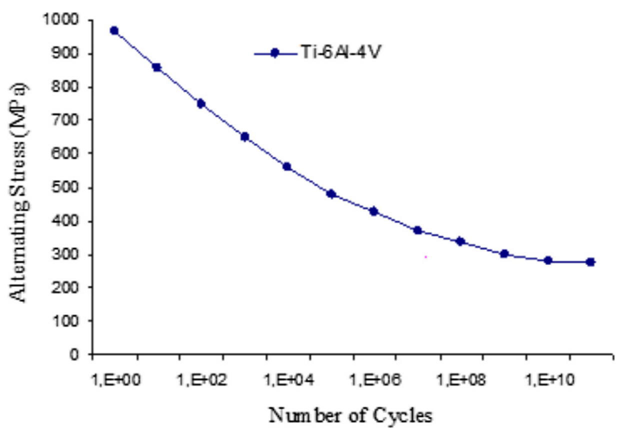


Fig. 4. Fatigue curves (S-N Curve) for Ti-6Al-4 V.

Table 6 shows that for all performance functions, deterministic shape optimization causes a reduction in the probability of failure. Compressive failure of dental implants and bone were reduced by 31.84% and 4.64%, respectively. When compared to the initial design, shear failure of the dental implant and the bone-dental implant interface was reduced by 33.59%, 28.60% and 44.61% respectively. Failure of dental implant fatigue and failure of bone fatigue decreased by 28.7% and 9.93% respectively for optimum dental shape. These results are very important in dental implant design. Implant fatigue is a very important parameter in terms of dental implant design criteria. Fatigue life of dental implant is increased while reducing stress on bone and dental implant during design.

Another important factor in dental implant design is to reduce stress in the bone and implant interface. Because the occurrence of more stress can damage the bone and lead to fracture of the bone. In dental implant design, dental implant-bone interface conditions are another factor to consider. In order to investigate these interfaces, 3 contact algorithms are generated to represent the interface states in the finite element model. When these 3 interfaces were examined, the bonded contact algorithm showed less stress than the others. As seen in Fig. 9, 18.22% reduction in dental implant-bone interface, 17.85% in dental implant and 17.25% reduction in bone.

A reduction in 28.92% of the deterministic response (reducing the stress on the dental implant with Ti-6Al-4 V material) reduced

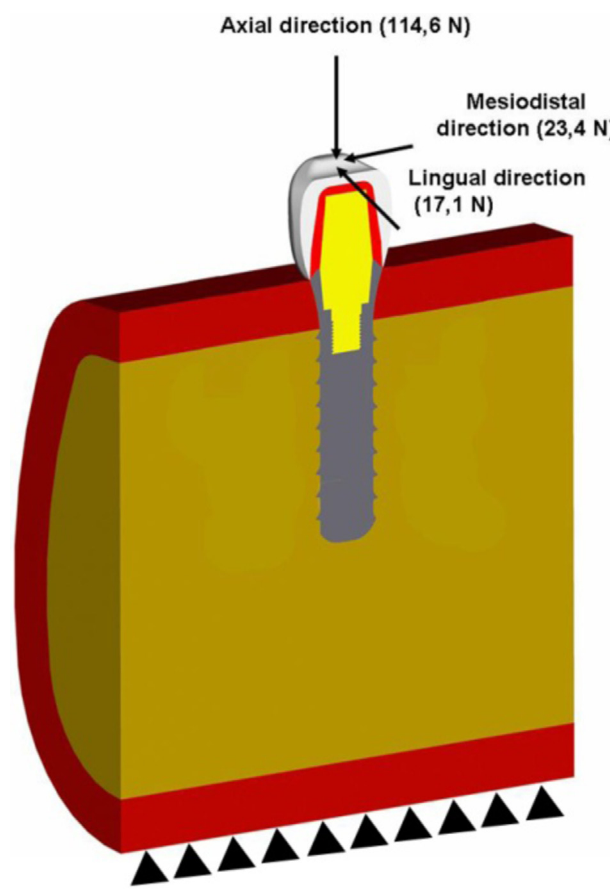


Fig. 5. Applied loads and boundary conditions of FEM model.

the likelihood of 31.84% compression failure, shear failure to 33.59% and fatigue error 28.7%. A reduction of 23.87% of the deterministic response (reducing stress in the bone) resulted in a failure of 4.64% compression failure with a failure rate of 28.6% and a reduction of fatigue of 9.93%. Dental implant shape optimization resulted in 19.27% reduction in von Mises stress in dental implant-bone interface, 23.48% in dental implant surface and 22.13% in bone. The results are shown in Fig. 10.

The axial load in the screw was determined versus axial displacement of the screw head measured contact area. The result

Please cite this article in press as: Kayabasi O. Design methodology for dental implant using approximate solution techniques. J Stomatol Oral Maxillofac Surg (2020), https://doi.org/10.1016/j.jormas.2020.01.003

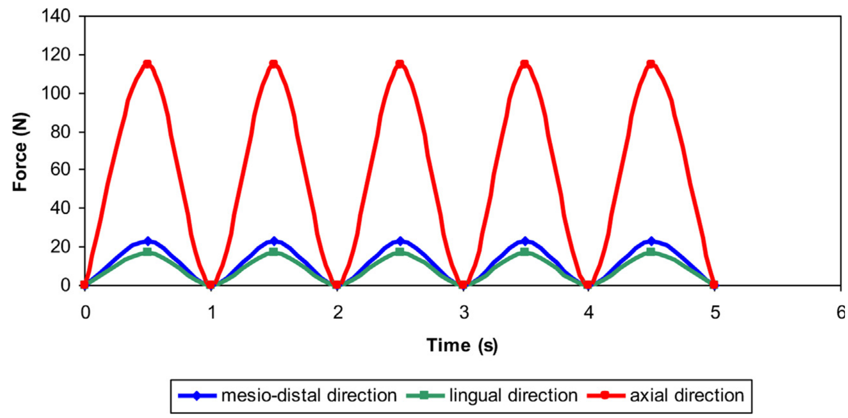


Fig. 6. Dynamic loading in five minutes.

Table 3
Manufacturer's recommended torque values.

Place of Application	Amount of Torque (Nmm) Mean	Amount of Torque (Nmm) S.D	Coefficient of variation (%)	Distribution Type
Bone-Implant Interface	3500	350	10	Lognormal
Abutment-Implant Interface	3500	350	10	Lognormal

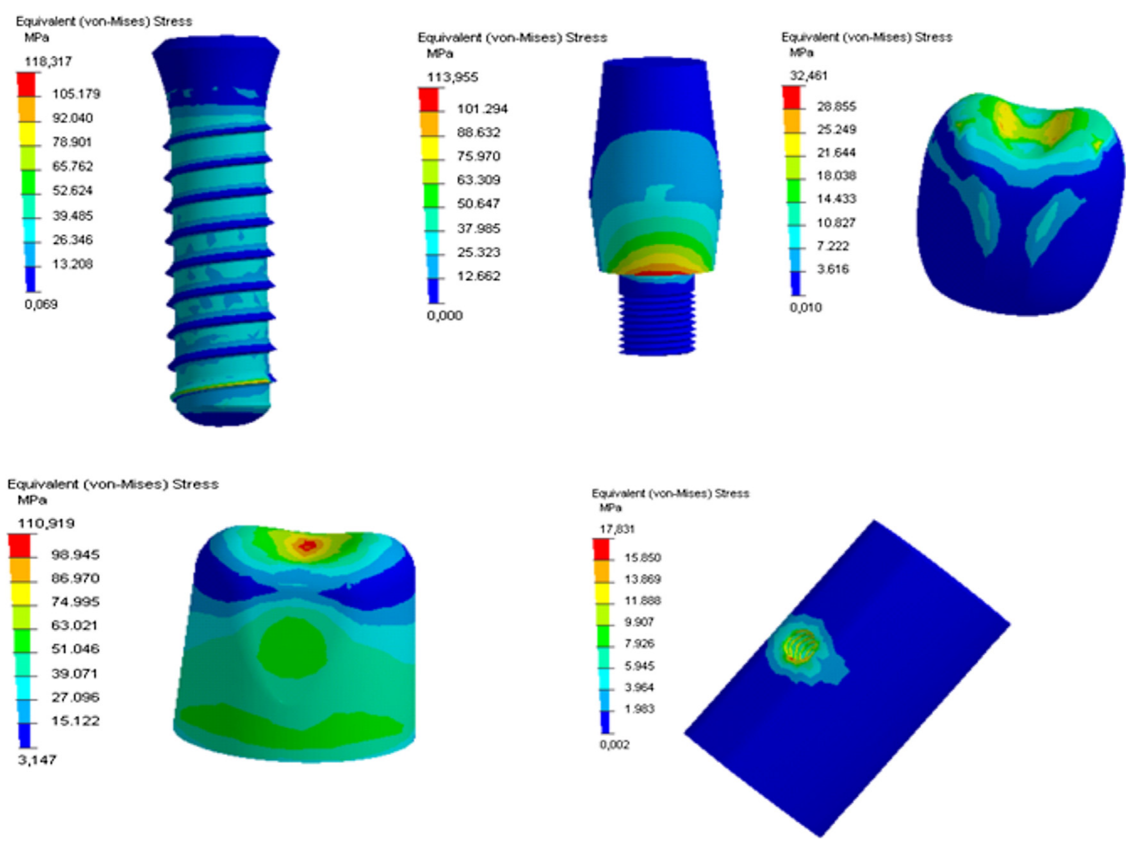


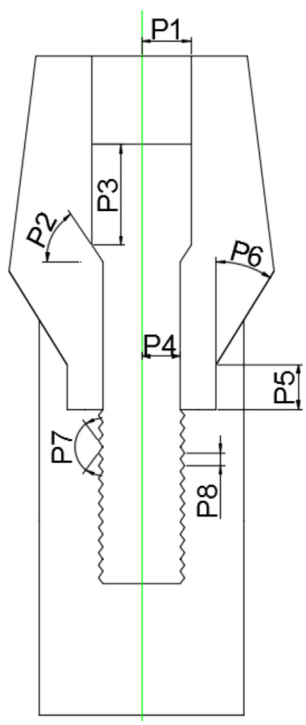
Fig. 7. Initial design results.

are expressed in Fig. 11. Clinically, axial preload is applied by torque. Within the elastic domain, a higher torque value shows nearly linear relationships between the axial displacement and clamping. In this study, the calculation of the pre-load dental implant design requirements in designing are showed how important it is. In optimum design has lower axial deformation than initial design.

The results are shown in Fig. 12 show that high levels of stress, with magnitudes up to ~ 16 MPa, exist in the bone around the neck of the implant. Optimum design shape has lower stress concentration than initial design shape. Theoretically, the stresses at a sharp corner or a point with material mismatch can be singular or infinite. The FE predicted that values for the stresses at these points would depend on the mesh density there. Mesh density gradually

Table 4
Performance function results.

Failure Criteria	Optimum Design p_f %	Initial Design p_f %	Improvement %
Compressive failure (Dental implant)	2.69	4.78	44
Shear failure (Dental Implant)	3.39	5.71	41
Fatigue failure (Dental Implant)	2.46	3.98	38
Compressive failure (Bone)	1.32	2.56	48
Shear failure (Bone)	1.69	3.09	45
Fatigue failure (Bone)	1.05	2.68	61
Tensile failure (Bone-Implant Interface)	2.56	4.96	48
Shear failure (Bone-Implant Interface)	1.78	4.21	58



P_1	1.7 mm
P_2	75°
P_3	3 mm
P_4	1.7 mm
P_5	2.8 mm
P_6	75°
P_7	60°
P_8	0.8 mm

Fig. 8. Optimum dental implant shape.

Table 5
Compared results.

	Optimum Design		Initial Design		Improvement %	
	Mean	S.D	Mean	S.D	Mean	S.D
Dental Implant von Mises stress (MPa) Ti-6Al-4V	95,21	12,69	118,02	16,87	23,96	32,94
Bone von Mises stress (MPa)	13,91	10,05	17,83	13,39	28,18	33,23
Interface von Mises stress (MPa) Dental Implant-Bone	15,67	6,52	20,51	9,16	30,89	40,49

Table 6
Performance function results.

Performance Function	Failure Criteria	Initial Design p_f (%)	Optimum Design p_f (%)	Change (%)
1	Dental implant compressive failure	29,37	43,09	31,84
2	Dental implant shear failure	19,39	29,2	33,59
3	Dental implant fatigue failure	29,53	41,42	28,7
4	Bone compressive failure	8,83	9,26	4,64
5	Bone shear failure	3,12	4,37	28,6
6	Bone fatigue failure	1,36	1,51	9,93
7	Bone-dental implant interface Tensile failure	51,9	78,26	33,68
8	Bone-dental implant interface shear failure	16,25	29,34	44,61

Please cite this article in press as: Kayabasi O. Design methodology for dental implant using approximate solution techniques. J Stomatol Oral Maxillofac Surg (2020), https://doi.org/10.1016/j.jormas.2020.01.003

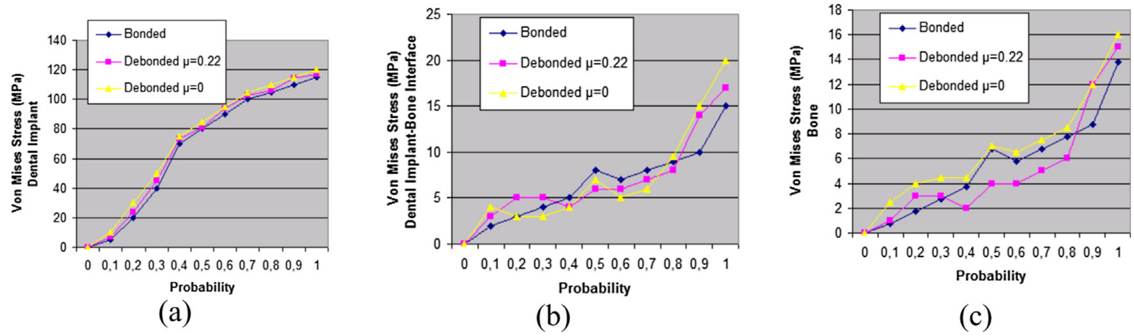


Fig. 9. Effect of deterministic design optimization. a. von Mises stress dental implant-bone interface. b. von Mises dental implant. c. von Mises stress bone.

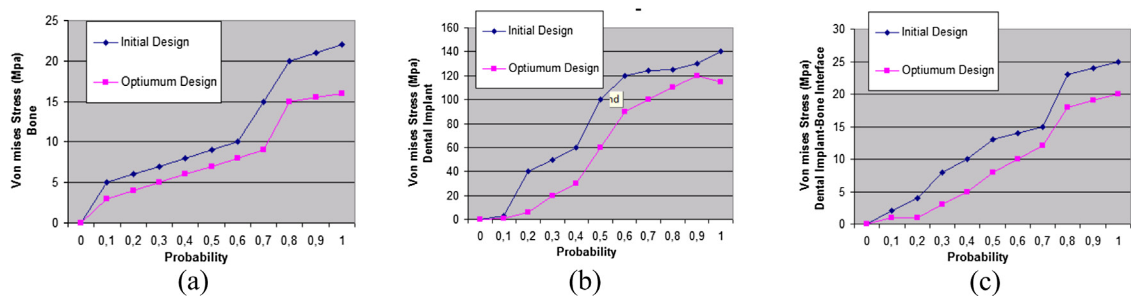


Fig. 10. Effect of deterministic design optimization. a. von Mises stress dental implant-bone interface. b. von Mises dental implant. c. von Mises stress bone.

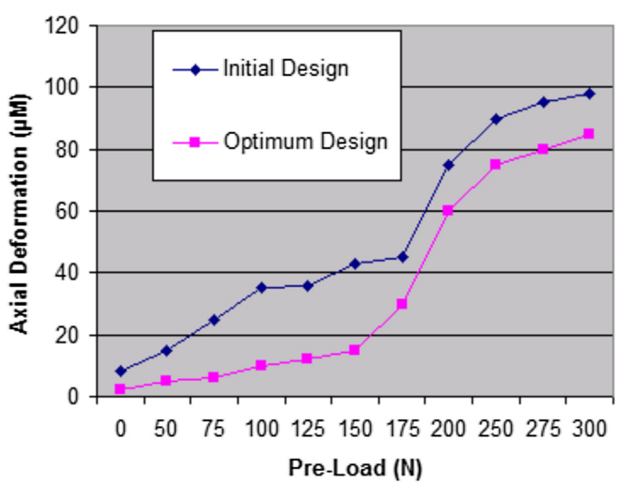


Fig. 11. The relationship between pre-load and axial deformation.

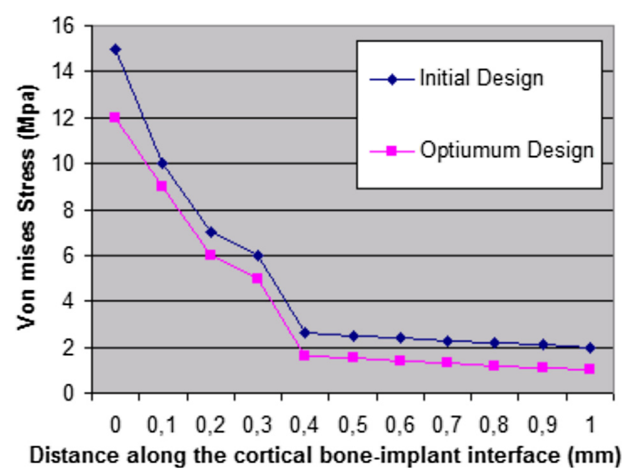


Fig. 12. Von Mises stress along the cortical bone–implant interface under axial loading.

coarsening from loading surfaces, which is potential impact region to, the bone and from contact surface to outer surface. Increasing the mesh density would lead to higher and higher predicted stresses at these points without ever achieving convergence. In order to make meaningful comparisons between different designs, therefore, the stresses along a length of the bone-implant interface in the vicinity of the stress concentration point were considered. This allowed comparison of the degrees of stress singularity, which is similar to the analysis of crack problems using fracture mechanics principles.

Prior to the fatigue analysis von Mises stresses obtained due to the applied loads were compared with the previous works to validate the model and to ensure the model safety against static failure. These values are under the yield strength value of the material. All the analysis were performed according to infinite life criteria (1e9 cycles). It is important that these maximum

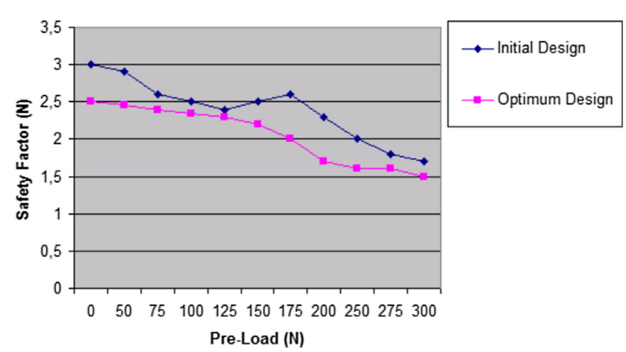


Fig. 13. Safety factor for Ti-6Al4 V initial design and optimum design.

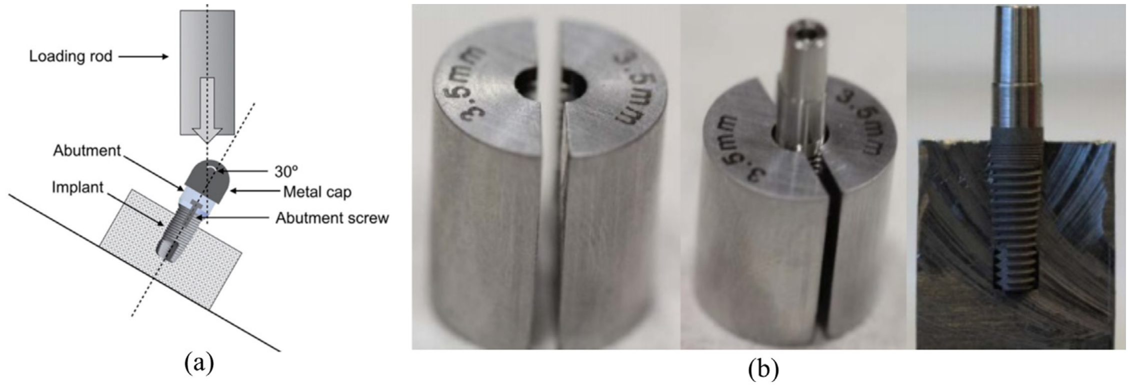


Fig. 14. Experimental setup.

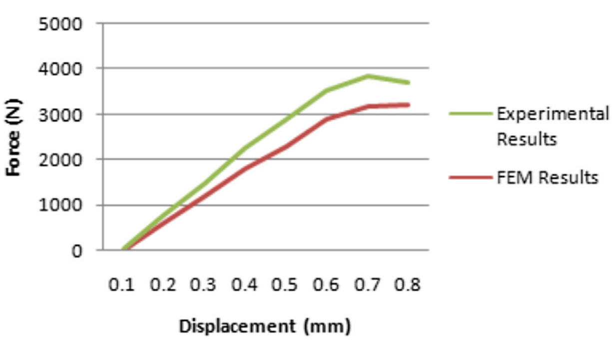


Fig. 15. Experimental and FEM results.

equivalent stress values should be lower than the endurance limit of the material. The endurance limits of Ti-6Al-4 V 138 MPa. Finite element analyses conducted in this study showed that implant geometry type is safe against fatigue load with Ti-6Al-4 V material. These results are shown in Fig. 13.

4. Experimental Setup

It is important that the dental implant has sufficient stability after insertion to the patient in order to provide optimal distribution of chewing and occlusal functional forces to the implant-bone interface after healing and to ensure the necessary and adequate bone formation around the implant during healing. Experimental setup was established to validate the results of the proposed design methodology. Firstly, the geometric shape which was obtained optimally was produced on CNC machine. The dental implant was placed with its own sleeves in a rigid-rigid 30-degree fixture. The fixture for static loading was placed on the Dartec brand tester. Experimental setup is shown in Fig. 14 (a) and (b). The compressive load was loaded to failure with a compression of 0.5 mm per minute between an unidirectional vertical platform and an angled sample. Applied force and displacement curve were recorded with computer software.

One of the limitations of this research is the selection of the optimal dental implant based on the dimensions of the designs of different companies. However, at the same time, all these implants have different internal connections that provide a broad assessment and expectation for the clinician. The main reason for this choice is the development of a new dental implant, eliminating the advantages and disadvantages of commercially available implants. Once we have achieved all these results, it is our main objective to compare them with other observation systems. The failure of the dental implant is screw joint instability, which involves loosening or breaking the abutments and screws. Relaxation and fracture are potential problems for all types of dental implants and screws. These problems usually arise from the design of the screw. Mechanical defects can be reduced by increasing the screw diameter of the dental implant and changing the screw joint design. The screw shape and dental implant body provide a large contact area between the implant and bone, increase stability, and reduce shear stress on the bone implant. The interface reduces stress concentration in the cervical region and alleviates stress concentration. Increasing the surface area of the implant improves the distribution of forces to the bone; therefore, various implant shapes have been developed to improve stability. Screw joint boot is one of the most important factors in dental implants. The boot reduces and prevents screw loosening. The boot should be as high as possible to maximize the contact force between the abutment and the implant. Preloading was used for both finite element analysis and experimental study as 3500Nmm. Compared experimental results and finite element analysis results, maximum flexibility gave very close results. The implant system showing maximum flexibility is presented in Fig. 15.

Fig. 16 shows the fracture dental implant. This result shows that the screw shape of the dental implant is an important parameter in the design.

Stress minimization is the most important parameter in the design of dental implants. In designs with high stresses, stresses

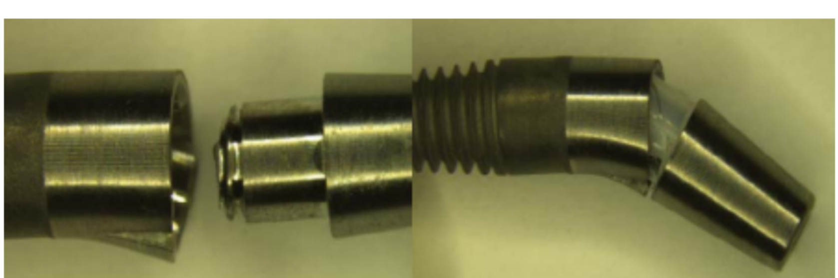


Fig. 16. Fracture dental implant.

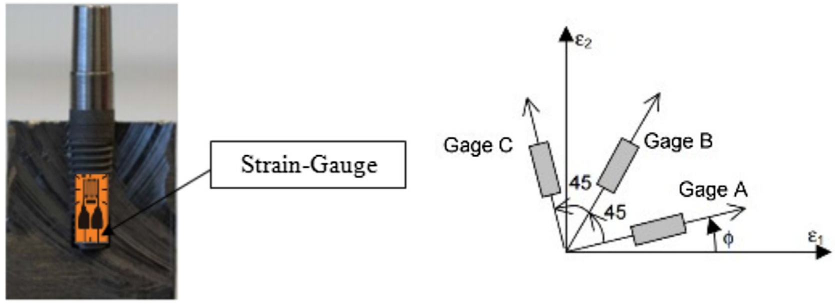


Fig. 17. Strain gauge placement.

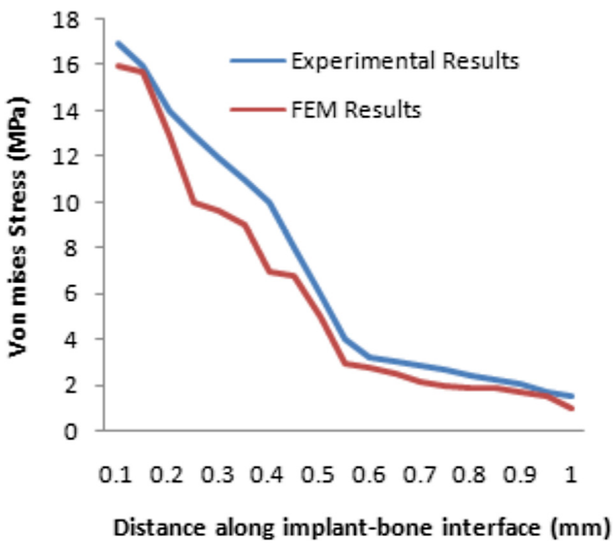


Fig. 18. Compared results Von misses stress.

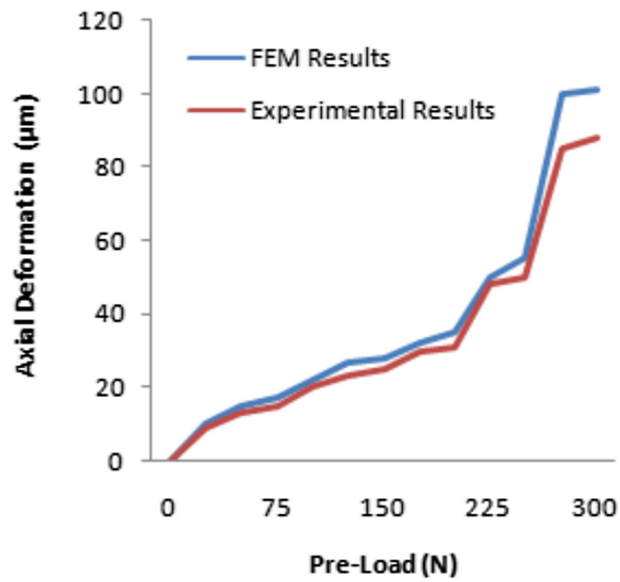


Fig. 19. Compared results axial deformation with pre-load.

will be transferred to the jawbone due to contact at the jawbone-dental implant interface. This will cause cracks in the jaw bone. Strain gauges were placed on the surface of the dental implant to test the accuracy of the proposed design methodology. In rosette strain gauges, the alignment can be at different angles. In this article, strain gauge arranged at 45° angles is examined. It is shown in Fig. 17. If the names of uniaxial gauges are called A, B and C respectively, the principal stresses are calculated using the following formulas:

$$\sigma_1 = E\left[\frac{\varepsilon_a + \varepsilon_c}{2(1-\upsilon)} + \frac{1}{2(1+\upsilon)}\sqrt{(\varepsilon_a - \varepsilon_c)^2 + (2\varepsilon_b - \varepsilon_a - \varepsilon_c)^2}\right] \quad (26)$$

$$\sigma_2 = E\left[\frac{\varepsilon_a + \varepsilon_c}{2(1-\upsilon)} + \frac{1}{2(1+\upsilon)}\sqrt{(\varepsilon_a - \varepsilon_c)^2 + (2\varepsilon_b - \varepsilon_a - \varepsilon_c)^2}\right] \quad (27)$$

$$\sigma = \sqrt{\sigma_1^2 + \sigma_2^2 - \sigma_1\sigma_2 + 2\tau_{12}^2} \quad (28)$$

Here σ1 and σ2 are principle stress. εa, εb, εc are strain values measured from the strain gauge and σ is Von misses stress. After measuring the strain values in the strain gauge, von Mises stresses in the dental implant were calculated using equations 25, 26 and 27. As seen in Fig. 18, the results obtained from the finite element analysis were found to be very convergent with the experimental results.

In addition, in order to simulate physical conditions, the dental implant was preloaded and experimentally measured in axial deformations. The comparative results of the results obtained with the finite element analysis are shown in Fig. 19.

5. Conclusion

The aim of this study is to use the optimization technique to obtain alternative shapes for dental implants in order to minimize stress distribution across the jaw-tooth implant interface. It is hoped that new designs will help to support and maintain the Osseo integration of dental implants. One of the most important factors in the design of dental implants is to investigate the dynamic and fatigue behaviour of dental implants. In this study, dynamic and fatigue behaviour of dental implants were investigated under different conditions. The design of the implant has been shown to withstand dynamic loading in all conditions at the end of the work. The designer uses reliability values according to customer demands. This leads to maximum safety and quality with minimum safety, so in this study, a probabilistic approach to three dimensional deterministic shape optimization of the dental implant was applied to influence the possibility of unpredictable implant system. Although the performances considered deterministically show a safe initial design for everyone other than dental implant-jaw interface failure, probabilistic analysis reveals the possibilities of finite failure. Probability analyzes allow the determination of the effect of uncertainty on system parameters. In the literature, many researchers have investigated the effects of static loading on dental implants. However, the specific variability and uncertainty of most of the major problem parameters were not considered. The aim of this study was to apply a computational approach to predicting the dental implant during fatigue life, considering fatigue strength, variability and uncertainty under loading conditions. A major limitation of numerical modeling in orthopedics is the inability to specifically take into account the uncertainty and variability associated with biological structures. In this study, uncertainty and variability are explained in the probabilistic finite element analysis of dental implant. Random variables are used to define joint and muscle loading, bone and dental implant material properties, and bone and interface forces. The risk of failure is clearly determined in terms of two failure modes: bone and bone implant interface failure. In each failure mode, various failure criteria are investigated. Optimal implant shapes using FE-based shape optimization techniques can potentially increase the success of dental implants due to low stress concentration at the bone implant interface, and all performance functions lower than the initial design are likely to fail. Optimal design dental implant results have better behavior characteristics than the initial design. In this study, boot and boot model is used to represent physical conditions. The results showed that the boot model had more stress than boot. These results showed that pre-implant should not be neglected in dental implant design. In order to estimate the behavior of dental implant systems, it is essential to understand and take into account the sources of diversity that the system or component under investigation will encounter. Very little has been done to reduce the effects of the stochastic structure of environment-mental variables on the performance of prosthetic systems. Another analysis investigated differences in environmental variables in dental implant design. In the present analysis, random variables are modeled more realistically as continuous functions, and a finite element model of the dental implant system is used to calculate the probabilistic response. The probability of failure was calculated and the effect of the deterministically controlled optimization on the calculated failure probability capability was monitored.

Experimental studies are very burdensome in terms of both cost and time. To reduce cost and time in the experimental study, the dental implant was originally designed in a computerized environment. For this purpose, the parametric model of the dental implant was created to obtain the optimum shape and the probability of failure was calculated using these results. After obtaining the most reliable result, the dental implant was experimentally tested to prove the accuracy of the results. When the results obtained were examined, it was seen that maximum stresses and deformation were in the neck region of the implant. This result showed that the design methodology we propose is that an implant can be designed in a computerized manner to give very realistic results.

The results obtained from finite element analysis and experimental results were very close to each other. The strategy we propose for the design methodology has been proven to be accurate. A general conclusion is that dental implants can be designed and operated with computer models before they are applied to the patient. This procedure helps prevent permanent damage from incorrect application and reduces design time.

Funding

There is no funding in this work.

Disclosure of interest

The author declare that he has no competing interest.

This article does not contain any studies with human participants performed by any of the authors. This article does not contain any studies with animals performed by any of the authors.

References

[1] Hong G, Rudi C, Van S, Newell J, Yew L. Dynamic modeling and simulation of dental implantin sertion process–A finite element study. Finite Elements in Analysis and Design 2011;47:886–97.
[2] Songhe L, Yongqiang Z, Chunlei L, Yingying S, Junrui Z, Dezhong X. Biomechanical optimization of the diameter of distraction screwing distraction implant by three-dimensional finite element analysis. Computers in Biology and Medicine 2013;43:1949–54.
[3] Sato Y, Wadamoto M, Tsuga K, Teixeira R. The effectiveness of element downsizing on a three-dimensional finite element model of bone trabeculae in implant biomechanics. Journal of Oral Rehabilation 1999;26:288–91.
[4] Sahin S, Cehreli C, Yalçın E. The influence of functional forces on the biomechanics of implant-supported prostheses-a review. Journal of Dentistry 2002;271–82.
[5] Chih C, Chen C, Chang H, Ming H. Finite element analysis of the dental implant using a topology optimization. Medical Engineering & Physics 2012;34:999–1008.
[6] Tao, Li, Kaijin H, Libo C, Yin D, Yuxiang D, et al. Optimum selection of the dental implant diameter and length in the posterior mandible with poor bone quality–A 3D finite element analysis. Applied Mathematical Modelling 2011;35:446–56.
[7] Dilek O, Tezulas E, Dincel M. Required minimum primary stability and torque values for immediate loading of mini dental implants: an experimental study in nonviable bovine femoral bone. Oral Surgery Oral Medicine Oral Pathology Oral Radiology Endodontogy 2008;105(2):20–7.
[8] Horiuchi K, Uchida H, Yamamoto K, Sugimur aM. Immediate loading of Brånemark system implants following placement in edentulous patients: a clinical report. International Journal of Oral Maxillofacial Implants 2000;6:824–30.
[9] Chiapasco M, Abati S, Romeo E, Vogel G. Implant-retained mandibular overdentures with Brånemark system MKII implants: a prospective comparative study between delayed and immediate loading. International Journal of Oral Maxillofacial 2001;4:537–46.
[10] Osterwyck H, Duyc J, Vander S, Vander G, Decoomans M, Lieven S, et al. The influence of bone mechanical properties and implant fixation upon bone loading around oral implants. Clinical Oral Implants Research 1998;6:407–12.
[11] Geng J, Tan C, Liu G. Application of finite element analysis in implant dentistry: a review of the literature. Journal of Prosthetic Dentistry 2001;6:585–98.
[12] Pietro A, Pasquale F, Massimo M, David W. Effects of thread features in osseo-integrated titanium implants using a statistics-based finite element method. Dental Materials 2012;8:919–27.
[13] Narendra KU, Arun TM, Nikhil I, Febin R. A 3D finite element analysis of dental implants with varying thread angles. Materials Today: Proceedings 2018;5:11900–5.
[14] Muhammad JH, Izzuddin Z, Safian S, Denni K, Fethma MN. Finite element analysis of mini implant biomechanics on peri-implant bone. Procedia Manufacturing 2019;30:308–14.
[15] Suchat A, Nantakrit Y, Nattawut J, Manutchanok J. Finite element analysis of dental implant prosthetics. Materials Today: Proceedings 2018;5:9525–34.
[16] Annike B, Meike S, Michael E, Andreas G. The effect of different occlusal contact situations on peri-implant bone stress–A contact finite element analysis of indirect axial loading. Materials Science & Engineering C 2019;99:367–73.
[17] Daniel P, Nicolella B, Tacke rH, Hamid K, Dwight D. The effect of the three-dimensional shape optimization on the probabilistic response of a cemented femoral hip prosthesis. Jorunal of Biomechanics 2006;39:1265–78.
[18] Oguz K, Bulent E. The Effects of Static Dynamic and Fatigue Behavior on Three-Dimensional Shape Optimization of Hip Prosthesis by Finite Element Method. Material & Design 2007;28:2269–77.
[19] ANSYS.. Ansys Theory Reference Manual Release 14. 0 Ansys Inc; 2014.
[20] Oğuz K, Emir Y, Fehmi E. Static, dynamic and fatigue behaviors of dental implant using finite element method. Advances in Engineering Software 2006;37:649–58.
[21] Alkan I, Sertgöz A, Ekici B. Influence of occlusal forces on stress distribution in preloaded dental implant screws. Journal of Prosthetic Dentistry 2004;91:319–25.