Frontiers in Dentistry

Finite Element Analysis of Stress in Bone and Abutment-Implant Interface under Static and Cyclic Loadings

Saeed Nokar[1], Hamid Jalali[1,2], Farideh Nozari[3], Mahnaz Arshad[2,4*]

1. Department of Prosthodontics, School of Dentistry, Tehran University of Medical Sciences, Tehran, Iran
2. Dental Research Center, Dentistry Research Institute, Tehran University of Medical Sciences, Tehran, Iran
3. Private Practice, Tehran, Iran
4. Department of Prosthodontics, School of Dentistry, Tehran University of Medical Sciences, Tehran, Iran

Article Info

Article type:
Original Article

Article History:
Received: 19 Dec 2019
Accepted: 25 Jul 2020
Published: 7 Sep 2020

**** Corresponding author:***
Dental Research Center, Dentistry Research Institute, Tehran University of Medical Sciences, Tehran, Iran

Email: mahnazarshad@yahoo.com

ABSTRACT

Objectives: The success of implant treatment depends on many factors affecting the bone-implant, implant-abutment, and abutment-prosthesis interfaces. Stress distribution in bone plays a major role in success/failure of dental implants. This study aimed to assess the pattern of stress distribution in bone and abutment-implant interface under static and cyclic loadings using finite element analysis (FEA).

Materials and Methods: In this study, ITI implants (4.1×12 mm) placed at the second premolar site with Synocta abutments and metal-ceramic crowns were simulated using SolidWorks 2007 and ABAQUS software. The bone-implant contact was assumed to be 100%. The abutments were tightened with 35 Ncm preload torque according to the manufacturer's instructions. Static and cyclic loads were applied in axial (116 Ncm), lingual (18 Ncm), and mesiodistal (24 Ncm) directions. The maximum von Mises stress and strain values were recorded.

Results: The maximum stress concentration was at the abutment neck during both static and cyclic loadings. Also, maximum stress concentration was observed in the cortical bone. The loading stress was higher in cyclic than static loading.

Conclusion: Within the limitations of this study, it can be concluded that the level of stress in single-unit implant restorations is within the tolerable range by bone.

Keywords: Finite Element Analysis; Fractures, Stress; Dental Implants; Bone

➢ **Cite this article as:** Nokar S, Jalali H, Nozari F, Arshad M. Finite Element Analysis of Stress in Bone and Abutment-Implant Interface under Static and Cyclic Loadings. *Front Dent.* 2020:17.21. doi: 10.18502/fid.v17i21.4315

INTRODUCTION

Several factors may affect the success of dental implantation; however, the reliability of the implant system itself is one of the topics of debate [1]. The implant design, quality and quantity of bone, surgical procedure, fit of restoration, proper occlusion, regular postoperative check-ups, and oral hygiene all affect the long-term success of dental implants [2]. The reliability of the implant system depends on two main factors namely the stress-bearing capacity and fatigue resistance [3]. Perfect design of implant leads to better stress distribution and fatigue safety during loadings [4]. The implant-abutment connection interface plays a pivotal role in reliability of the implant system [5].

Stress distribution at the dental implant-cortical bone interface is a major concern for dental clinicians. Attempts have been made to decrease stress in cortical bone and achieve a stress pattern similar to that around natural teeth [6]. Dental implants and their components must resist masticatory forces, which have a cyclic pattern [7]. Most stress analyses have been performed using finite element method under static conditions, and the analysis of cyclic response of dental implants has not been performed with finite element method [8, 9]. The aim of this study was to evaluate the stress induced in the implant abutment and the surrounding bone under different cyclic loads by finite element analysis (FEA).

MATERIALS AND METHODS

To simulate the components in FEA, the components needed to be measured first. The dimensions of a 12×1.4 mm ITI implant (Straumann, Switzerland) and 5.5 mm ITI regular neck solid abutment (Straumann, Switzerland) (RN) were measured using OMT Toolmakers' microscope (Optical Measuring Tools Ltd., UK) with 0.005 mm accuracy and a profile projector (Inspection Enlarger, Hull, Yorkshire, UK) with 0.01 mm accuracy. According to the measurements, the dimensions of the components were estimated as follows:

(I) Fixture: 12×1.4 mm standard ITI implant with 1.25 mm distance between the threads and thread depth of 0.35 mm. The internal diameter of the implant fixture was 3.5 mm at the platform and had 8° taper. There was an octagon with 1.26 mm sides inside the implant body. The implant screw hole had 3.4 mm distance from the platform with M2×0.4 threads.

(II) ITI regular neck solid abutment had 5.5 mm length with an aperture/apical screw with five M2×0.4 standard threads.

After measuring the components, the measurements were plotted as two-dimensional maps in AutoCAD software. Next, SolidWorks 2006 software (Dassault Systems, Massachusetts, USA) was used to model the components.

A segment of mandible at the site of an extracted second premolar was modeled. The bone type was simulated to be D2 according to the Lekholm and Zarb's classification [10], and 2 mm of cortical bone surrounded the cancellous bone.

The bone was modeled as a block measuring 25×23 mm, and the fixture was placed at the center of bone segment. The bone thickness was 1.37 mm at the buccal plate and 1.37 mm at the lingual plate relative to the implant. To design the superstructure, an actual tooth model was used. Computed tomography scans were used for modeling of teeth such that the scans were uploaded to Mimics software to yield 3D model of the tooth. The designed 3D model was then transferred to FEA software. Chromium-cobalt metal frameworks and metal-ceramic restorations with feldspathic porcelain were then designed.

Mechanical properties:

The mechanical properties of the materials used were defined based on the type of analysis. In this study, static, dynamic, and implant fatigue analyses were performed.

The purpose of static analysis is to predict the behavior of a model in response to different loading conditions independent of time. The properties of the material in static analysis are as follows:

1. Modulus of elasticity, the Poisson's coefficient.
2. For inertial loads (such as weight and angular velocity), the density of the material should also be determined.
3. In case of thermal loading, the coefficient of thermal expansion must also be specified.

Dynamic analysis is performed to estimate the dynamic response of a model under the influence of time-dependent loading. In this analysis, we can calculate the displacements, strains, stresses, and time-dependent variations in loads in a model. Fatigue analysis was carried out according to Goodman, Soderberg and Gerber's formulas. Mechanical properties required to calculate fatigue according to Goodman, Soderberg and Gerber's formulas are yield stress (Sy), ultimate stress (Su), and tensile strength (Se).

Goodman's formula: $\left(\frac{\sigma_a}{s_e}\right) + \left(\frac{\sigma_m}{s_u}\right) = \frac{1}{N}$

Soderberg's formula: $\left(\frac{\sigma_a}{s_e}\right) + \left(\frac{\sigma_m}{s_y}\right) = \frac{1}{N}$

Gerber's formula: $\left(\frac{N\sigma_a}{s_e}\right) + \left(\frac{N\sigma_m}{s_u}\right)^2 = 1$

Table 1 shows the mechanical properties of the materials used in this study. All materials were considered isotropic, homogeneous and linearly elastic. Ti-6Al-4V (UNS designation

R56400), also sometimes called TC4, is an alpha-beta titanium alloy with a high strength-to-weight ratio and excellent corrosion resistance. It is one of the most commonly used titanium alloys and is applied in a wide range of applications where low density and excellent corrosion resistance are necessary e.g. aerospace industry and biomechanical applications (implants and prostheses).

Table 1. Mechanical properties of the materials used in this study

Component	Material	E	V	YS
Implant	Ti-6Al-4v	105	0.33	800
Abutment	Ti-6Al-4v	105	0.33	800
Framework	Cr-Co alloy	220	0.30	720
Porcelain	Feldspathic	61.2	0.19	500
Cortical bone	Bone	14.8	0.30	130
Cancellous bone	Bone	1.85	0.30	130

E: Young's modulus (Gpa); V: Poisson's ratio; YS: Yield strength

Modeling contact:
The interaction between bone and implant during dynamic simulation of the implantation process is complex and requires definition of contact conditions. In the present study, contact was defined in ABAQUS (2007) using surface-to-surface discretization because it enables more accurate stress and load distribution than node-to-surface discretization.

Implant-abutment assembly:
After determining the mechanical properties of the components, the components were placed in the software environment. The fixture was first placed in bone (fixture collar was located outside the bone). Then, the solid abutment was tightened to the fixture. The abutment was attached to the implant with a tightening torque of 35 Ncm [11] (Fig. 1a). In this type of contact, the two objects start to slip on each other when the load exceeds a certain threshold (Fig. 1b).

Meshing:
Meshing was performed using ABAQUS 6.9 software (Dassault Systems, Massachusetts, USA). Tetrahedron C3D4 elements were used in this study. The number of elements and nodes for different components is presented in Table 2.

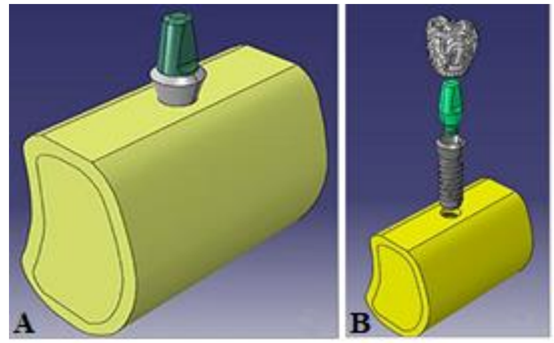

Fig. 1. (A) Implant-abutment assembly; (B) load and boundary conditions of the 3D finite element analysis model

Table 2. Element (C3D4) and node numbers in meshing of components

Component	Elements (N)	Nodes (N)
Implant	32410	7083
Solid abutment	52405	10101
Cancellous bone	17163	3540
Cortical bone	10217	2741

Boundary and loading conditions:
In this study, cyclic loads were applied to crown in axial, lingual, and mesiodistal directions. Maximum load was 116 N in axial, 18 N in lingual, and 24 N in mesiodistal direction to simulate three meals a day, each lasting for 15 min.
Averagely 60 masticatory cycles occur per minute, which would be 2700 cycles per day and one million cycles per year. Boundary conditions were defined such that the model could not move under loading. The bone segment was fixed from the mesial and distal, and the loads were applied to the center of crown. Time-dependent masticatory loads were applied. Dynamic and cyclic loads (500,000, 1 million, and 3 million cycles) were applied for 60 s. Stress levels were calculated and reported as von Mises stress values.

RESULTS

The maximum von Mises stress was 10.2 MPa in bone, 139 MPa in implant and 102 MPa in abutment. Table 3 shows the maximum von

Mises stress values in bone, implant, and abutment under different loading cycles. The location of maximum stress concentration under static and cyclic loadings was in the coronal region of bone, implant neck, and abutment shank.

Table 3. Maximum stress in different loading cycles

Cycles	Bone	Implant	Abutment
500,000	10.4	151	144
1 million	11.9	158	152
3 million	12.83	162	156

Figure 2 shows the value of N for all three conditions. In all three conditions, the value of N for both the implant and the abutment was higher than one, indicating that the abutment and the implant under study did not experience fatigue.

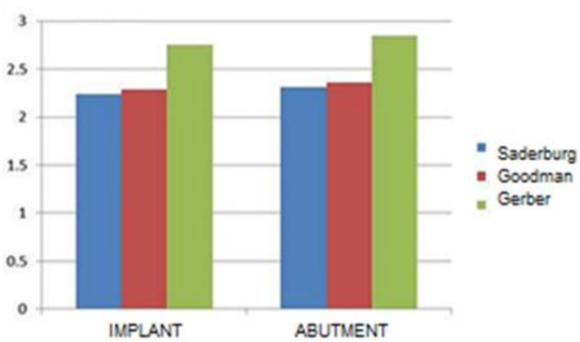

Fig. 2. N value for all three conditions. There are three computational charts that show little difference which was not significant

According to Figure 3, the maximum and minimum stress values in the implant after eliminating the extreme data were 158 and 25 MPa, respectively. Due to the possibility of error, we did not consider the stresses created in the initial cycles and did not include them in our calculations. The Qm of implant was 91.5. The Qa of implant was calculated to be 66.5, the Se was equal to 200, and the Su was equal to 870 for titanium. Putting these values in the formula, the value of N was 2.284, which indicates that the implant used under loading does not suffer from fatigue failure and has an unlimited lifespan. The maximum and minimum stress values in the abutment were 152 and 23, respectively. As a result, the Qm and Qa were 87.5 and 64.5, respectively. By placing these numbers in the formula N, we obtained 2.363, showing that the abutment did not experience fatigue in loading. Similarly, in the Saderberg's and Gerber's formulas, we put the numbers and obtained the value of N.

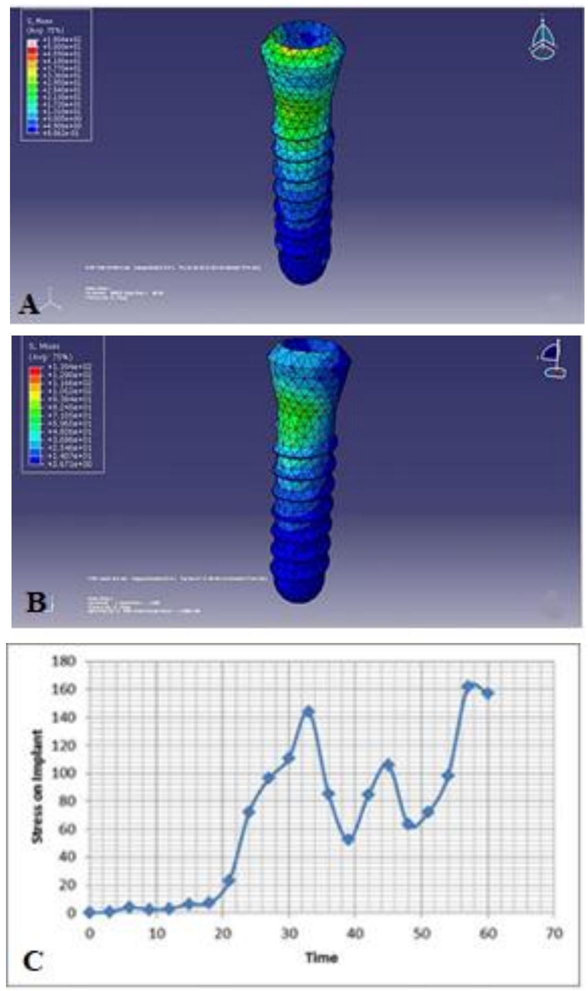

Fig. 3. (A) Stress distribution pattern in implant during cyclic loading; (B) stress distribution pattern in implant during dynamic loading; (C) stress pattern in implant during 60 s of masticatory cycle

Implant:

Figures 3a and 3b show the pattern of stress distribution in the implant under static and cyclic loads. The maximum amount of stress was concentrated at the implant neck in both types of implant loading. Figure 3c shows the stress pattern in implant during 60 s of masticatory cycles. The maximum von Mises stress in implant under static and cyclic loadings was much lower than the yield stress (yield point for the Ti-6Al-4V alloy is 800 Mpa).

Abutment:

Figures 4a and 4b show the pattern of stress distribution in the abutment under static and cyclic loadings. The maximum amount of stress was concentrated at the abutment shank under both types of loadings. Figure 4c shows the pattern of stress distribution in the abutment under masticatory cycles for 60 s. The maximum amount of stress in the abutment was 12.7% and 19.5% of the yield stress under static loading and cyclic loading with 3 million cycles, respectively. Thus, maximum von Mises stress values under both static and cyclic loadings were much lower than the yield stress.

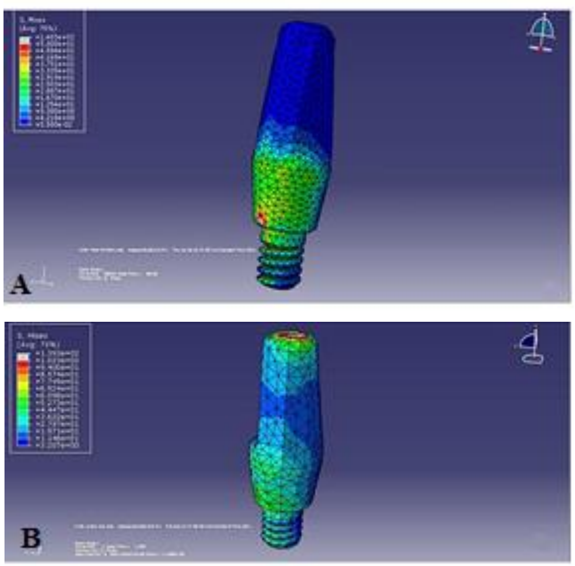

Fig. 4. (a) Stress distribution pattern in abutment during static loading; (b) stress distribution pattern in abutment during cyclic loading; (c) stress pattern in abutment during 60 s of masticatory cycles

Bone:

Figures 5a and 5b show the maximum stress in bone under static and cyclic loadings. The maximum amount of stress was concentrated at the bone crest at the level of implant neck and in cortical bone under both types of loadings. Under static loading and cyclic loading with 3 million cycles, the maximum amount of stress in bone was 7.6% and 9.8% of the yield stress, respectively. Thus, the maximum von Mises stress values in both static and cyclic loadings were much lower than the yield stress (yield point for bone is 130 MPa).

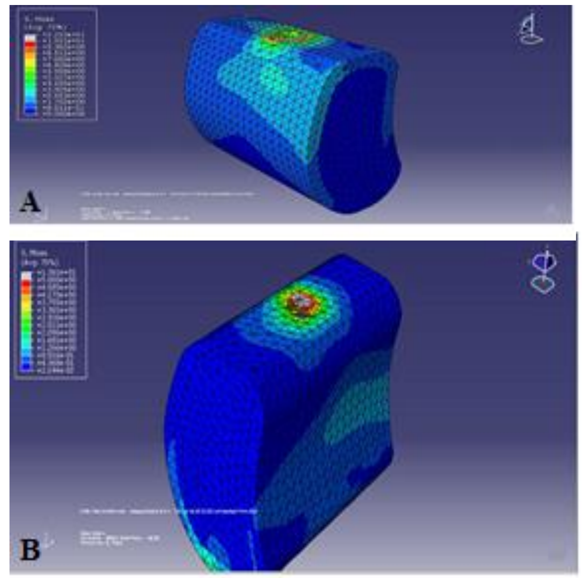

Fig. 5. (a) Stress distribution pattern in bone during static loading; (b) stress distribution pattern in bone during cycling loading

DISCUSSION

Natural teeth may be lost due to severe caries, periodontal disease or trauma, and dental implants are the most efficient treatment option for replacement of the lost teeth [12,13]. Dental implants are subjected to different loading cycles during mastication, which may result in their failure [14]. In dental implants, stress is often accumulated at the implant-bone interface, and plays a key role in success/failure of dental implants [15]. Fatigue life depends on several factors including implant itself, physical properties of bone, and other characteristics of occlusion.

FEA is a valuable method to study the pattern of stress distribution in implant and peri-implant bone [16]. This analysis can help in assessment of stress level at the bone-implant interface to ensure long-term success of dental implants in the clinical setting [9]. The biomechanics of the implant-screw-abutment complex is complicated.

As the concept of osseointegration began to consolidate, clinicians paid more attention to the implant-restoration interface and the related problems such as screw loosening, screw fracture, restoration fracture, and even implant fracture due to the increasing prevalence of such complications compared with problems related to poor osseointegration [9]. Application of excessive occlusal loads to implant restoration can result in stress accumulation at the bone-implant interface, and consequent crestal bone resorption around dental implants. As the result, implant mobility and its subsequent failure are likely to occur [17]. In earlier screw designs, the abutment screw loosening had a reported frequency of 6% while single crown loosening had 25% prevalence [17, 18]. The greater the height of crown connected to the abutment, the greater the level of stress applied to the screw would be, which would increase the risk of screw loosening [19]. Minatel et al. [20] discussed that it is important to know the stress distribution pattern in dental implants because it allows us to predict where the fracture or failure would occur.

The Ti-6Al-4V titanium alloy is a biomaterial used for the fabrication of dental implants. As shown in the current study, the maximum stress was concentrated at the abutment neck during both static and cyclic loadings. Also, the maximum stress concentration occurred in cortical bone. The loading stress was higher in cyclic than static loading. Perez [21] studied dental implants placed in the second premolar region of a human mandible and modeled them using Mimics software. Semi-automatic segmentation was performed for the bone segment in their study to differentiate between cortical and trabecular bone. Based on their findings, implant failure occurred at the implant neck. Another study reported that the maximum concentration of stress was at the abutment-screw-implant connection. Failure most commonly occurred at the upper screw threads, due to high-stress concentration in this area. Maximum stress was found to be concentrated at the first implant thread under both static and cyclic loadings [22]. The maximum bite force by the stomatognathic system has been recorded in the posterior region in the range of 300-800 N at the site of first molar [23]. The maximum stress was 62.51 MPa for the titanium structure under vertical loading, 244.52 MPa for the titanium structure under oblique loading, 64.02 MPa for the zirconia framework under vertical loading, and 229.74 MPa for the zirconia framework under oblique loading. In the oral cavity, the applied loads are mostly dynamic in nature; thus, evaluation of loading conditions by static loading leads to simplification of the complex biomechanical processes that take place in the oral cavity [24]. With the introduction of cyclic loading, there is no significant change in the magnitude of stress or strain. This may be attributed to the fact that there are no cumulative stresses transferred to the cortical bone, which may show an increase in stress magnitude over time. Hence, the magnitude of stress and strain remains the same in cyclic loading [6]. However; other factors may influence the obtained results. During the fracture process, stress concentrates at the tip of the cracks, and thus, accelerates the crack propagation. Finally, the material fractures as the concentrated stress exceeds a certain threshold [25]. For dental implants, the loading angle is also an important parameter in fracture analysis. The pattern of stress distribution in implants may change as the result of alteration in loading angle [25].

In conclusion, within the limitations of this study, it can be concluded that the stress value in bone around single-unit implant restorations is within the tolerable range. Stress distribution and implant stability in osteoporotic bone are more sensitive to implant design compared with normal bone. By preventing implant overload and ensuring adequate primary stability, we can promote a safe biomechanical environment for implant survival.

CONCLUSION

The strain rate was about 650 microspheres under static loading and 800 microspheres under cyclic loading in our study, which would not cause fatigue failure. According to Frost [26], 1-2 MPa stress causes 50-100 strains in mammalian bones. In this study, the maximum stress in static loading was 10 MPa and the maximum stress in cyclic loading was 12 MPa. Due to the fact that the stress created in non-isotropic bone is 20-30% more than in isotropic bone, and considering the amount of strain to be 650 microstrains in static loading and 800 microstrains in cyclic loading, these values are in the range of 50-1500 microstrains, and do not cause fatigue failure.

CONFLICT OF INTEREST STATEMENT

None declared.

REFERENCES

1. Zhang X, Liu L, Wang Y, Chen X. Effects of dental implant-abutment interfaces on the reliability of implant systems. MATEC Web Conf., 2016 Oct;77:08006.
2. Lee JI, Lee Y, Kim NY, Kim YL, Cho HW. A photoelastic stress analysis of screw- and cement-retained implant prostheses with marginal gaps. Clin Implant Dent Relat Res. 2013 Oct;15(5):735-49.
3. Mehdi G, Belarbi A, Mansouri B, Azari Z. Numerical study of effect of elastomeric stress absorbers on stress reduction in bone-dental implant interface. J Appl Oral Sci. 2015 Jan-Feb;23(1):87-93.
4. Zhang X, Xie ZG, Feng W, Chen XS, Chen JY. Effects of custom-made and thread dental implant system on the stress distribution in alveolar bone: a 3-dimensional finite element analysis. Appl Mech Mater. 2014;475:1487-93.
5. El-Anwar MI, Yousief SA, Soliman TA, Saleh MM, Omar WS. A finite element study on stress distribution of two different attachment designs under implant supported overdenture. Saudi Dent J. 2015 Oct;27(4):201-7.
6. Mishra M, Ozawa S, Masuda T, Yoshioka F, Tanaka Y. Finite element study on the effect of abutment length and material on implant bone interface against dynamic loading. J Adv Prosthodont. 2011 Sep;3(3):140-4.
7. Prados-Privado M, Bea JA, Rojo R, Gehrke SA, Calvo-Guirado JL, Prados-Frutos JC. A new model to study fatigue in dental implants based on probabilistic finite elements and cumulative damage model. Appl Bionics Biomech. 2017;2017:3726361.
8. Gehrke SA. Importance of crown height ratios in dental implants on the fracture strength of different connection designs: An in vitro study. Clin Implant Dent Relat Res. 2015 Aug;17(4):790-7.
9. Segundo RM, Oshima HM, da Silva IN, Burnett LH Jr, Mota EG, Silva LL. Stress distribution of an internal connection implant prostheses set: a 3D finite element analysis. Stomatologija. 2009;11(2):55-9.
10. Lekholm U, Zarb GA. In: Patient selection and preparation. Tissue integrated prostheses: osseointegration in clinical dentistry. Branemark PI, Zarb GA, Albrektsson T, editor. Chicago: Quintessence Publishing Company; 1985. p. 199–209.
11. Kayabasi O, Yuzbasioglu E, Erzincanli F. Static, dynamic and fatigue behaviors of dental implant using finite element method. Adv Eng Softw. 2006 Oct;37(10):649-58.
12. Quek HC, Tan KB, Nicholls JI. Load fatigue performance of four implant-abutment interface designs: effect of torque level and implant system. Int J Oral Maxillofac Implants. 2008 Mar-Apr;23(2):253-62.
13. Dhatrak P, Shirsat UM, Deshmukh V. Fatigue life prediction of commercial dental implants based on biomechanical parameters: A review. IJSEIMS. 2015 May;3(2):221-6.
14. Chang CL, Chen CS, Yeung TC, Hsu ML. Biomechanical effect of a zirconia dental implant-crown system: a three-dimensional finite element analysis. Int J Oral Maxillofac Implants. 2012 Jul-Aug;27(4):49-57.
15. Hou PJ, Ou KL, Wang CC, Huang CF, Ruslin M, Sugiatno E, et al. Hybrid micro/nanostructural surface offering improved stress distribution and enhanced osseointegration properties of the biomedical titanium implant. J Mech Behav Biomed Mater. 2018 Mar;79:173-180.
16. Pierrisnard L, Renouard F, Renault P, Barquins M. Influence of implant length and bicortical anchorage on implant stress distribution. Clin Implant Dent Relat Res. 2003 Dec;5(4):254-62.
17. Tian K, Chen J, Han L, Yang J, Huang W, Wu D. Angled abutments result in increased or decreased stress on surrounding bone of single-unit dental implants: a finite element analysis. Med Eng Phys. 2012 Dec;34(10):1526-31.
18. Saab XE, Griggs JA, Powers JM, Engelmeier RL. Effect of abutment angulation on the strain on the bone around an implant in the anterior maxilla: a finite element study. J Prosthet Dent. 2007 Feb;97(2):85-92.
19. Brozović J, Demoli N, Farkaš N, Sušić M, Alar Z, Gabrić Pandurić D. Properties of axially loaded implant-abutment assemblies using digital

holographic interferometry analysis. Dent Mater. 2014 Mar;30(3):17-27.
20. Minatel L, Verri FR, Kudo GAH, de Faria Almeida DA, de Souza Batista VE, Lemos CAA, et al. Effect of different types of prosthetic platforms on stress-distribution in dental implant-supported prostheses. Mater Sci Eng C Mater Biol Appl. 2017 Feb;71:35-42.
21. Pérez MA. Life prediction of different commercial dental implants as influence by uncertainties in their fatigue material properties and loading conditions. Comput Methods Programs Biomed. 2012 Dec;108(3):1277-86.
22. Shemtov-Yona K, Rittel D. Identification of failure mechanisms in retrieved fractured dental implants. Eng Failure Anal. 2014 Mar;38:58-65.
23. Belli S, Eskitaşcioğlu G, Eraslan O, Senawongse P, Tagami J. Effect of hybrid layer on stress distribution in a premolar tooth restored with composite or ceramic inlay: an FEM study. J Biomed Mater Res B Appl Biomater. 2005 Aug;74(2):665-8.
24. https://www.infona.pl/resource/bwmeta1.element.elsevier-39e6659f-9c7d-3d18-8a75-c5d6efe20a15
25. Huang HM, Tsai CM, Chang CC, Lin CT, Lee SY. Evaluation of loading conditions on fatigue-failed implants by fracture surface analysis. Int J Oral Maxillofac Implants. 2005 Nov-Dec;20(6):854-9.
26. Frost HM. A 2003 update of bone physiology and Wolff's Law for clinicians. The Angle Orthod. 2004 Feb;74(1):3-15.