Uludağ University Journal of The Faculty of Engineering, Vol. 23, No. 3, 2018 RESEARCH

DOI: 10.17482/uumfd.299899

EFFECT OF DENTAL IMPLANT DIMENSIONS ON FATIGUE BEHAVIOUR: A NUMERICAL APPROACH

Hüsna TOPKAYA *

Mete Onur KAMAN **

Received: 24.03.2017; revised: 14.09.2018; accepted: 30.11.2018

Abstract: In this study, the effect of the implant shape on fatigue behavior was investigated with the finite element method. The implant material used was Ti6Al4V. The examination focus on the effect on the implant life with respect to changes in the crest and root dimension, thread depth, thread pitch, implant length, implant diameter, chamfer length, chamfer radius, and groove length to implant length ratio. Models were subjected to fatigue analysis according to ISO 14801 standard with ANSYS finite element software. The mean stress correction theory is chosen in fatigue life solutions. As a result, the parameters seen to have the most effect on dental implant fatigue behavior were the implant length and diameter. The parameters with the most effect on the implant screw fatigue were pitch and the height of the tooth.

Keywords: Dental implant, Fatigue, Finite element method, ISO 14801.

Dental İmplant Ölçülerinin Yorulma Davranışına Etkisi: Sayısal Bir Yaklaşım

Öz: Bu çalışmada implant şeklinin yorulma davranışına etkisi sonlu elemanlar metoduyla araştırılmıştır. İmplant malzemesi olarak Ti6Al4V kullanılmıştır. Çalışmada diş dibi kalınlığı, diş üstü kalınlığı, diş yüksekliği, diş hatvesi, implant boyu, implant çapı, pah boyu, pahın yarıçapı ve oyuk boyu değerlerinin değişiminin implant ömrüne etkisi araştırılmıştır. Modeller ISO 14801 standardında belirtilen kriterlere göre ANSYS programında yorulma analizine tabi tutulmuştur. Sonuçta dental implantların yorulma davranışını en çok etkileyen boyut değerlerinin implant çapı ve implant boyu olduğu görülmüştür. İmplant vidası yorulma ömrü üzerindeki en etkili parametreler hatve ve diş yüksekliğidir.

Anahtar Kelimeler: Dental İmplant, Yorulma, Sonlu Elemanlar Metodu, ISO 14801.

1. INTRODUCTION

Dental implants are used throughout the day and are exposed not only to damage by static force, but also to dynamic loading during chewing. Besides dynamic loading, bone quality, implant design, the structure of the implant surface and clinical applications all affect the success of implant treatment (Dilek et al., 2008). In addition, as chewing forces vary between genders and age groups, the loads affecting implants and dental prosthesis also show variations (Daas et al., 2008). Therefore, awareness of the fact that implant life depends on the conditions of use is of great importance. At this point, there is a need to select the appropriate geometry and materials associated with the fatigue life for the ideal dental implant application.

Experimental methods are important techniques used for the definition of the fatigue life of implant systems. The *ISO 14801* standard is used to examine fatigue behaviour of dental

* Batman Üniversitesi Batman MYO Makine Programı 72080 Batman

** Fırat Üniversitesi Mühendislik Fakültesi Makine Mühendisliği Bölümü 23200 Elazığ

Correspondence Author: Hüsna TOPKAYA (husna.topkaya@batman.edu.tr)

implants (ISO 14801 2007). This standard includes detailed information on how the load is applied to the implant, the position of the implant, and test procedures. In a study by Silva (Silva et al. 2008), fatigue experiments were conducted on forty eight ceramic implants. No procedure was applied to twenty four, while the other twenty four were covered with an acrylic resin crown according to the *ISO 14801* standard. A load of 600 N at 50000 cycles was applied with a *Weibull* curve and at the end of the study there was shown to be no difference between the two groups.

Dittmer (Dittmer et al. 2011) compared the load–bearing capacity and the failure resistance of six different forms of implant-abutment connection. As a result of the fatigue experiments applied according to *ISO 14801*, it was seen that the form of the implant-abutment connection had an important effect on load-bearing capacity and the damage created on the implant. Park (Park et al. 2008) investigated the fatigue behavior of five different combinations of implant and abutment. The fatigue resistance of zirconium abutment was found to be higher than that of the titanium abutment. In studies which have investigated fatigue behavior of dental implants experimentally, it has been reported that a decrease in lingual components of the forces affecting dental implants increased implant stability and therefore the life of the implant (Barbier et al. 1998, Moraes et al. 2015, Figueiredo et al. 2014).

In parallel with current developments in technology, there is increasing use of numerical methods to define the stress in dental implant systems. One of the most widely-used of these methods is the finite element method (Meriç et al., 2011, Şahin et al., 2002). As a result of force applied to the implant, stress can be calculated with the finite element method, and the fatigue life and failure can be examined. In this method, volumes are divided into small parts and stress and strain values are calculated more easily. Then, with the totals of the stress and strain of the small areas, the values for the whole are calculated (Ekici, 2002).

Lin (Lin et al. 2010) made a numerical examination of a 2-Dimensional model of the stress distribution occurring in the cortical and trabecular bone, and as a result of using Functionally Graded Material (FGM) and titanium as implant material, it was determined that stress was distributed in a more homogenous form by FGM. It was also shown in the study that the placement of FGM had better compatibility with the bone tissue, which then recovered more quickly. In another study, Djebbar (Djebbar et al. 2010) examined stress distribution on dental prostheses using the finite element method. The effects on stress of a change in load intensity and direction were investigated and it was demonstrated that an increase in the lingual component of loading increased stress on the implant and led to more intense stress in the area where the implant and abutment connected. Tao Li (Li et al. 2011) applied stress analysis to define the optimum dimensions of implant diameter and length for a weak jaw bone using a 3-dimensional finite element method on implants 3-5 mm in diameter and 6-16 mm in length. It was determined that under occlusal loading, increasing diameter and length decreased *von Mises* stress on the cortical and trabecular bone at a rate of 76 % and the most suitable implant diameter and length was determined to be 4 mm and 12 mm, respectively.

In a study by Ausiello (Ausiello et al. 2012), the pitch, width, angle and thickness values of the tooth in the implant geometry were changed in sixteen different implant models and the degree of bone damage under 350N loading was examined using the finite element method. Karl and Kelly (Karl and Kelly 2009) examined fatigue behaviour for sixty six implant samples by applying loading of 20-420 N and 20-500 N at 10^6 cycles at 2 Hz and 30 Hz frequencies to acrylic, glass-fiber, reinforced epoxy resin and aluminum materials according to the *ISO14801* standard. The study's results showed that at 420 N damage was caused to thirteen of the eighteen samples at 2 Hz and six of the eighteen samples at 30 Hz.

Sevilla (Sevilla et al. 2010) applied fatigue analysis according to *ISO 14801* to two groups of zirconium implant samples, one group with a 4 mm diameter, 11 mm length and 6 mm crest module and the other group with a 3.8 mm diameter, 10 mm length and 4.2 mm crest module. Kayabasi (Kayabasi et al. 2006) examined the static and dynamic fatigue behaviour of dental implants. Dynamic loading was applied to the occlusal surface for 5 minutes and the fatigue life was calculated according to the *Goodman*, *Soderberg*, *Gerber* and *Mean Stress* theories. Schiefer (Schiefer et al. 2009) designed implant systems with *CATIA v5* software and applied fatigue analysis according to *ISO 14801* using *ANSYS Workbench* software.

There are increasing numbers of numerical studies on stress distribution in dental implants under static and dynamic loading. Research done on fatigue life has generally been an examination from the experimental aspect with some new numerical applications (Geringer et al. 2014, Prados-Privado et al. 2015). There are many studies on the effects of dental implant dimensions on stress and fatigue distributions of dental implants. The most important contribution of the current study to the literature is the investigation of the effects of whole dental implant design parameters on stress and fatigue life. In the current study, fatigue life was researched numerically with the effects of repeated forces on an implant system and the effect was examined on the implant life of a change in the dimension parameters of the implant model. The implant system was created with the *SolidWorks* software, imported to *ANSYS Workbench* software, then fatigue analysis was applied with the stress-life method approach. To define fatigue life, dental-implant system and loading conditions were modeled according to *ISO 14801*. The effects of the implant dimensions on fatigue behaviour were examined and the results were presented in graphic form.

2. MATERIALS AND METHODS

A typical dental implant system consists of three main parts: the implant, the abutment, and the screw that holds the implant and abutment together. The abutment is used to create a connection between the implant, which is placed into the bone, and the upper structure (framework, prosthesis etc.), which is placed on the implant.

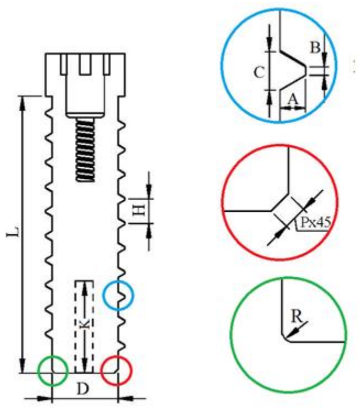

Figure 1.
Implant dimensions.

The aim of the study was to investigate the effect of dental implant dimensions on fatigue life numerically. Therefore, models of different measurements were created and the analysis was repeated for each model. Two basic changes were taken as the basis for the models. The first of these was the dimensional properties of length, diameter and groove length of the implant (L, D, K,) and the second was the thread dimensions, thread pitch, and the length and radius of the chamfer (A,B,C, H, P, R) (Figure 1).

The implant dimensions used in the analysis are shown in Figure 1 and Table 1. Each time only one of these dimensions was changed, the analysis was repeated and the failure loads and fatigue lives were calculated for the models. The properties sought in implant materials are primarily that it does not harm the bone tissue and is biologically compatible. Thus, titanium alloys are the leading materials in the manufacture of dental implants. In the current study, the titanium alloy *Ti6A14V* was selected as the material for the implant, abutment and for the screw used to fix the implant to the abutment.

Table 1. The dimensions of the implant used in the analyses.

Dimensions	(mm)			
A	0.10	0.20	0.30(Yang and Xiang, 2007)	0.40
B	0.10(Yang and Xiang, 2007)	0.15	0.30	0.45
C	0.10	0.15	0.30	0.45(Yang and Xiang, 2007)
D	3.00	3.10(Yang and Xiang, 2007)	3.20	3.30
H	0.85	1.00	1.15(Yang and Xiang, 2007)	1.30
K/L	0.00	0.30	0.40	0.50
L	12.00	13.00(Yang and Xiang, 2007)	14.00	15.00
P	0.00	0.20×45°(Yang and Xiang, 2007)	0.40×45°	0.60×45°
R	0.00	0.50	1.00	1.55

The Young's Modulus of *Ti6A14V* was 110000 MPa, yield stress 880 MPa and Poisson's ratio was 0.35 (Kayabasi et al. 2006). The trabecular and cortical bone, which function as the jawbone and to which the implant system is connected, were modeled separately. The hard and resistant section of cortical bone and the trabecular bone, which remains inside, were accepted as linear isotropic. For these bones, the Young's moduli were 13700 MPa and 1370 MPa, respectively and Poisson's ratio was 0.3 (Kitagawa et al. 2005). As the bone/ implant system was connected, the substructure was defined as a rigid material.

The fatigue analyses were applied with the approach of the stress life method. According to this method, the load is applied to the values which will not exceed the yield stress of the material being tested. The stress-life method is also known as high cycle fatigue with cycles selected in the range 10^3- 10^6. Stress-life is concerned with total life and does not distinguish between initiation and propagation. Since the fatigue analysis applied in the study was on a stress basis, the definition of the stress-cycle number (S-N) of the implant material graph was sufficient.

The graph (S-N) used in the fatigue analyses is shown in Figure 2-c. During the fatigue analyses, loading was applied according to the *ISO 14801* standard. The implant was tested by attaching a hemispherical load part to it. The load was applied to the implant axis at an inclination of below 30° and the implant was fixed 3 mm from the nominal bone level. The centre of the load element in hemispherical form is defined for the loading point to be 11 mm from the level of the connection to the bone. The load application and the boundary conditions are shown in Figure 2-a and 2-b (l=11 mm, y=4 mm) (Topkaya, 2014). While conducting the fatigue analysis $R=\sigma_{min}/\sigma_{max}$ was selected as 0.1.

The different implant systems, with the dimensions given in Table 1, were created with *SolidWorks* software. Next, the solid model was imported into *ANSYS Workbench* software. The connection type between interfaces of the implant components was defined as bonded. At this point the solid model formed and the material properties defined were meshed. Tetrahedron element type was used for meshing the parts. A tetrahedron has ten nodes and each of these has six degrees of freedom and can be used for isotropic materials.

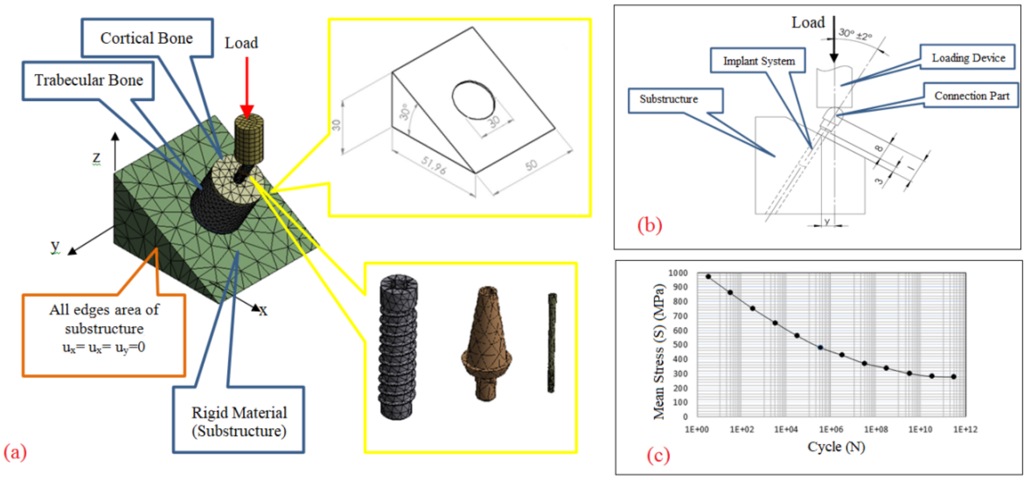

Figure 2.
a-Finite element model and boundary conditions of the numerical analysis study, b-Geometrical dimensions of dental implant system (ISO 14801 2007), c-The S-N graph for the Ti6Al4V material (Lin et al., 2010).

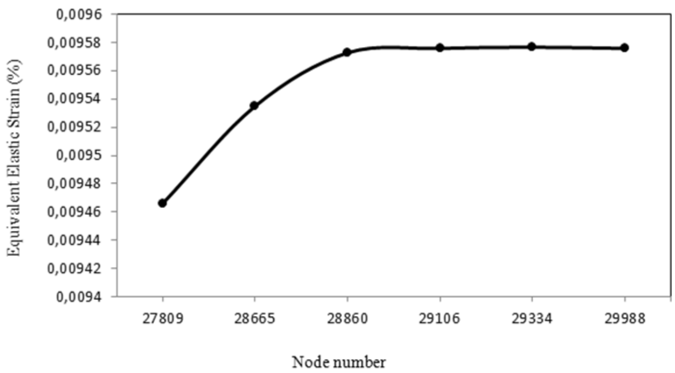

Figure 3.
The change in strain with number of nodes.

The analyses were applied to determine the optimum number of nodes and elements in the created mesh structure. A change was observed with the number of nodes of the strain in the whole of the created model. The starting point of the horizontal course of the curve gives the most appropriate number of nodes. For the models used in the analyses for this study, the most appropriate number of nodes was in the range of 28860-29000 (Figure 3). The meshed implant system and the boundary conditions are shown in Figure 2-a.

In summary, process steps of numerical fatigue analysis for dental implant applications can be listed as follows;

- *Step-1*: Numerical, three dimensional models for dental implants are prepared in SolidWorks and then converted to the *ANSYS Workbench* finite element program.
- *Step-2*: Material properties (Young's Modulus, Poisson's ratio, yield stress) are entered into the program. The model is meshed using tetrahedrons finite elements.

- *Step-3*: Boundary conditions (all edges, area of substructure, u_x= u_x= u_y=0) and loads are applied to the numerical model. Bonded type contact properties are chosen for all parts of dental implant automatically.
- *Step-4*: The numerical problem solved statically and stress and deformation values are obtained.
- *Step-5*: In this step, fatigue analysis begins. The loading frequency is selected as 10 Hz and the constant amplitude load option is R=σ_{min}/σ_{max}=0.1, as fully reversed. *Mean stress* correction theory is chosen for equivalent (*von Mises*) stress component in fatigue life solutions. Fatigue strength factor (K_f) is using for reducing the fatigue strength of test sample due to harsh conditions etc. (K_f) is taken as 1.0 for all analysis. Next, fatigue tool box solves the problem according to the *von Mises* stress values in the nodes which are obtained from static analysis.
- *Step-6*: As a result, fatigue life results are obtained numerically. Initially, random large force values are applied to the head of the implant in order to obtain the finite fatigue life value. Then, force values are decreased for every numerical fatigue solution, until infinite fatigue life is obtained.

3. RESULTS AND DISCUSSION

In this study, a dental implant system made with *Ti6A14V* material was examined with the finite element method according to the fatigue analysis found in the *ISO 14801* standard. The effect that the implant dimensions had on fatigue life was investigated. The analysis applied in the *ANSYS Workbench* program was applied in two stages. The first stage was static analysis and the second stage was the determination of the fatigue life.

Initially, for validating the method an experimental fatigue investigation and a static stress problem of dental implant systems, which was solved in literature, were examined and the results were compared.

Problem dimensions and boundary conditions can be obtained from References 4 and 28. *Ti6Al4V* was used as the implant material and the mechanical properties of all the materials were as specified in the materials and methods section (Figure 4). Evaluation was made of the *von Mises* stress distribution created in the implant according to the different loading angles for the modeled implant system. The maximum *von Mises* stress values, obtained in accordance with literature for loading angles $\theta = 0^o$ and 15^o, are shown in Table 2. The nearest studies were chosen for validating the current study. Especially Yang and Xiang's study and Kaman et al.'s study has same boundary conditions and geometrical values. The main difference between Chun et al.'s study and current study is implant length. While 13 mm implant length was chosen in current study, Chun et al. selected as 10 mm length.

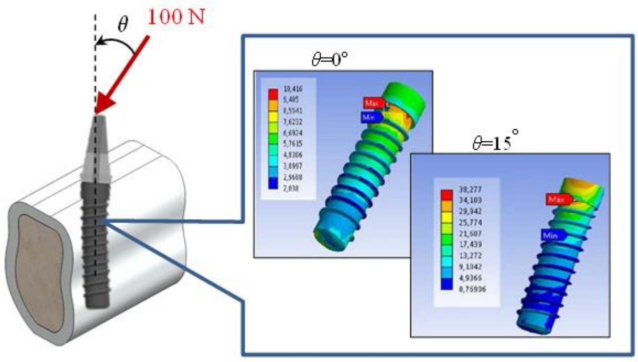

Figure 4.
The implant problem associated with the trabecular and cortical bone system loaded with a 100 N single load (Yang and Xiang et al., 2007, Chun et al., 2002).

Table 2. Comparison of the static stress results of the current study and previous studies in literature.

θ=0º		θ =15º	
Study	Maximum *von Mises* stress (MPa)	Study	Maximum *von Mises* stress (MPa)
Yang and Xiang 2007 (3D)	10.580	Yang and Xiang 2007 (3D)	41.034
Chun et al. 2002 (2D)	14.461	Chun et al. 2002 (2D)	29.926
Kaman et al. 2012 (3D)	12.437	Kaman et al. 2012 (3D)	39.709
Current study (3D)	12.416	Current study (3D)	38.277

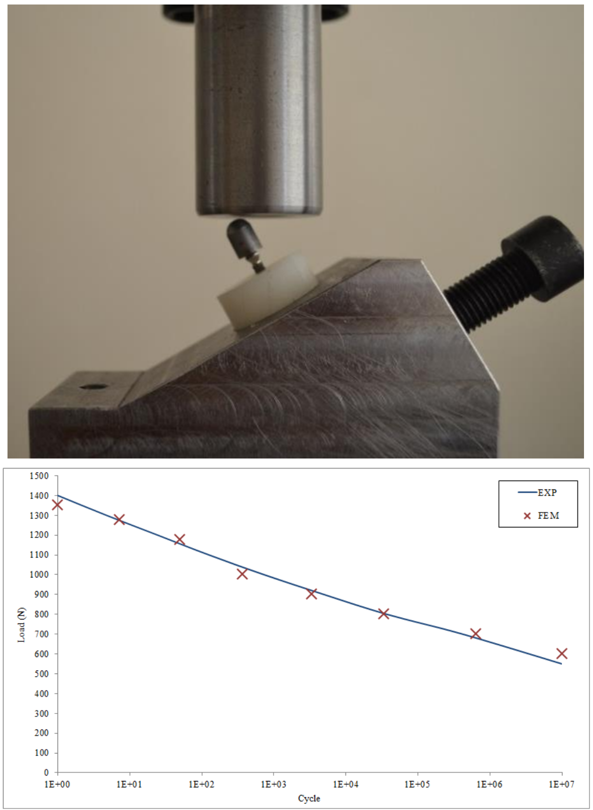

Figure 5.
Experimental validation a- experimental setup, b-comparison of FEM results with experimental results.

For experimental validation fatigue experiments applied to Tekka brand 3.1 mm diameter and 13 mm length dental implant. Fatigue tests were made with Shimadzu brand servo hydrolic test equipment. Test frequency was selected as 10 Hz and R=0.1. Experimental setup and comparison with finite element analysis is given in Figure 5.

Damage starts to occur in the implant from the fatigue that begins from the loading. Together with the emergence of damage, the safety factor falls below the value of 1. In the fatigue analysis made using the data given for (Yang and Xiang, 2007) in Table 1, the minimum life occurred at the first contact point of the implant with the cortical bone. The point at which fatigue starts in the implant is the same point at which damage starts.

By just changing one of the dimensions given as (Yang and Xiang, 2007) in Table 1, and holding the other parameters fixed, the loads were defined which started fatigue in the models and the fatigue life values were calculated. The graphs showing the load changes applied with the fatigue life for the changes in different implant dimension parameters are shown in Figures 5-6.

The effect of changes in thread depth on implant life are shown in Figure 6-a. The smallest value of A, 0.1mm, was taken at the lowest value of implant life. With an increase in the value of A, the implant life increased and the highest implant life was seen to be 0.4 mm for A in the model. The fatigue limit was taken as the value 430 N in the model, where A was 0.1 mm. As A increased, so did the fatigue limit. For the models at 0.2 mm, 0.3 mm and 0.4 mm, the fatigue limit was 460 N, 505 N and 520 N, respectively. A change in the dimension of the crest was seen to have less effect than the thread depth (Figure 6-b). For the same load values, the model with B of 0.45 mm had higher fatigue life than the other models. In all the applied loads, the implant life and fatigue limit increased with an increase in B.

Fatigue analysis was made for four different values of the helical thread root used to fix the implant to the bone (Figure 6-c). An increase in the value of C caused an increase in the implant life. While the fatigue limit was 465 N for C=0.1 mm, at C=0.45 mm this value was 505 N.

When the graphs of Figure 6 are examined, it can be seen that the parameter which changed fatigue behaviour the most was thread depth. Ao (Ao et al., 2010) reported that thread depth played a more significant role than the length of crest in terms of the stress created in the bone and abutment implant stability.

At the highest pitch value of 1.3 mm, the minimum implant fatigue life occurs. A large thread pitch value means that less of the implant will be held by the screw tooth side in the jawbone. A reduction in pitch increases the implant strength to fatigue (Figure 5-d). The highest fatigue life value was seen in the model with pitch of 0.85 mm. Kong (Kong et al., 2009) reported optimum pitch length of 0.8 mm in respect of stress created in the bone and implant abutment stability. At large pitch values (H=1.3 mm) at the fatigue limit, sudden drops are observed and H becomes as important as thread depth in fatigue life.

For the same load value, when implant diameter, D, is reduced, a higher level of stress is expected to be created. At the lowest value of implant diameter, at 3 mm, the fatigue life of the implant was lower than at other diameter values (Figure 7-a). As the implant diameter increased, the fatigue life of the implant increased. At the same time, although the rate of increase of the diameter was the same for each model, the fatigue values became closer to each other. The highest fatigue life values were observed in the model with the implant diameter of 3.3 mm. Li (Li et al., 2011) showed that with increased implant diameter, the stress created in the bone decreased. In the *ISO 14801* standard, when the implant is placed with a 30° vertical angle, the increased implant length (L) will create an amount of a moment and thus by increasing the amount of stress when the implant length is increased, the implant life was decreased (Figure 7-b).

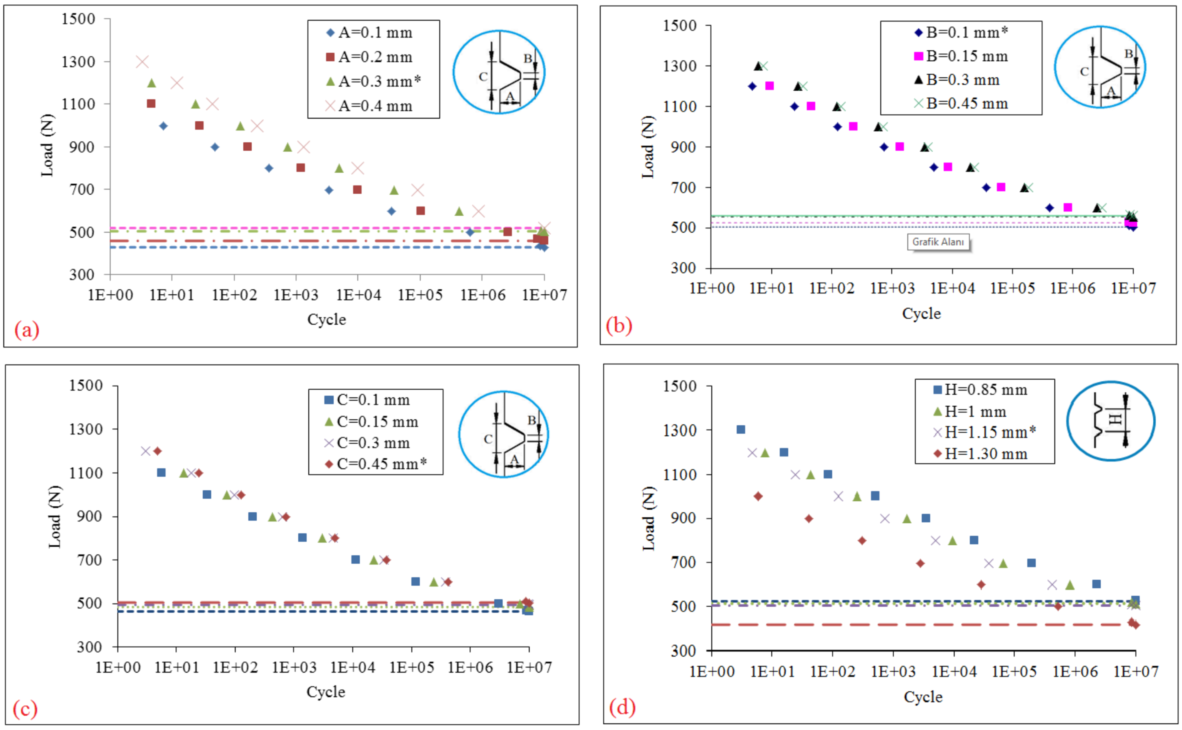

Figure 6.
The changes in fatigue life with the load applied for different a-implant thread depth (A) values, b-implant thread crest length (B) values, c-implant tooth root thickness (C) values, d-implant thread pitch (H) values.

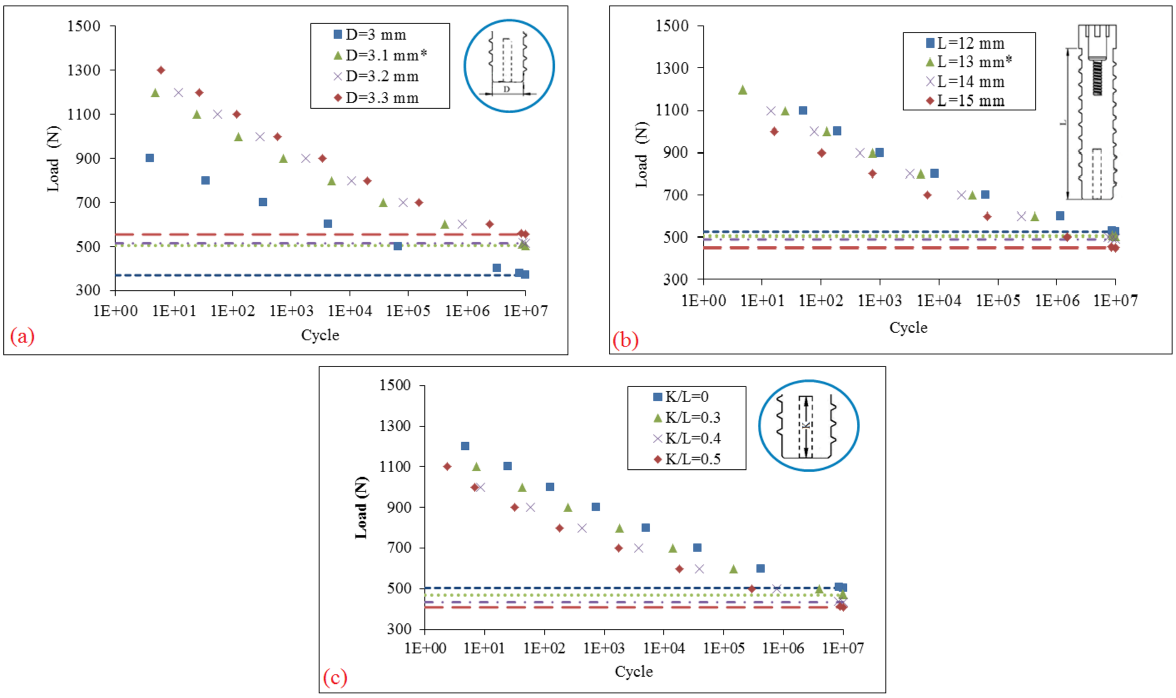

Figure 7.
The change in fatigue life with the load applied for different a-implant diameter (D) values, b-implant length (L) values, c-groove length (K) values.

The lowest fatigue life values were observed in the model with an implant length of 15mm, and as implant length decreased, fatigue life increased. In clinical applications, when it is necessary to have vertical placement of implants, the likelihood of creating a moment is very low. However, because of the torque which could be created from non-occlusal forces on vertically placed implants that are not placed vertically, the lowest possible length value would be more useful in terms of implant life. Additionally, the parameters specified for the implant diameter and length conforms to the structure of the jaw of the patient. Therefore selection of the ideal implant for the patient will be the largest diameter possible, which will then prolong the implant life.

In order for the implant to be held by the jawbone, there should be as much osseointegration as possible. The use of different implant materials, different surface coatings, different surface smoothness values and expanding the grooves at different lengths of the implant can all be applied to help increase osseointegration. As a result of the analyses it was seen that expanding the grooves to increase osseointegration between implant and bone decreased implant life (Figure 7-c). The implant life values against the applied load were highest in the ungrooved model. It was determined that as the length of the opened grooves increased, the implant life decreased.

4. CONCLUSIONS

In this study, a numerical examination was made of the fatigue behaviour of dental implants. Fatigue analyses were applied to models created in the *SolidWorks* program using *ANSYS Workbench* software according to the criteria specified in the *ISO14801* standard. When creating the implant models three things were investigated: the changes that can be made by implant manufacturers to create different implant models, the dimension parameters and the effect of these differences on fatigue behaviour. Evaluations, made in the light of the results obtained for each of the above parameters, are provided below:

- When considering the helical screw used to fix the implant to the jawbone, an increase in the thread depth, the length of crest and thread root thickness increases implant life and an increase in pitch causes a decrease in implant life.
- An examination of the analyses applied in this study showed that a decrease in implant diameter, and an increase in implant length and groove length deceased implant life.
- The most effective of the implant screw parameters on fatigue life were pitch and thread depth. Conversely, the least effective were thread root thickness and crest length. When the effects of implant diameter and length on fatigue life were compared, diameter was seen to be a more effective parameter than length. The length of the grooves to be opened in the implant is an important variable with as much effect on fatigue life as the diameter. There are some limitations, such as full-bonded contact between implant components, stress ratio, parallel force to the implant axis et cetera, in this numerical solution that affects real implant life. The ideal implant dimensions should be selected by reviewing the clinical applications under optimum conditions.

REFERENCES

1. Ao J., Li T., Liu Y., Ding Y., Wu G., Hua K., Kong L. Optimal design of thread height and width on an immediately loaded cylinder implant: A finite element analysis, Computers in Biology and Medicine 2010; 40: 681-686 *doi:10.1016/j.compbiomed.2009.10.007.*

2. Ausiello P., Franciosa P., Martorelli M.,Watts D.C. Effects of thread features in osseo-integrated titanium implants using a statistics-based finite element method. Dental Materials 2012; 28: 919-927. *doi: 10.1016/j.dental.2012.04.035*

3. Barbier L., Vander Sloten J., Krzesinski G., Schepers E., Van Der Perre G. Finite element analysis of non-axial versus axial loading of oral implants in the mandible of the dog. Journal of Oral Rehabilitation.1998;25:847-858.*doi: 10.1046/j.13652842.1998.00318.x*

4. Chun H-J., S.-Y. Cheong S-Y., Han J-H., Heo S-J., Chung J-P., Rhyu I-C., Choı Y-C., Baik H-K., Ku Y. ve Kım M-H. Evaluation of design parameters of osseointegrated dental implants using finite element analysis. Journal of Oral Rehabilitation 2002; 29: 565–574. *doi: 10.1046/j.1365-2842.2002.00891.x*

5. Daas, M., Dubois, G., Bonnet, A. S., Lipinski, P., Rignon-Bret, C. A complete finite element 224 model of a mandibular implant retained overdenture with two implants: Comparison between rigid and 225 resilient attachment configurations. Medical Engineering & Physics 2008; 30: 218-225. *doi: 10.1016/j.medengphy.2007.02.005*

6. Dilek O., Tezulas E., Dincel M. Required minimum primary stability and torque values for immediate loading of mini dentalimplants: an experimentalstudy in nonviable bovine femoral bone. Oral Surgery, Oral Medicine, Oral Pathology, Oral Radiology, and Endodontology 2008; 105: 20-27. *doi: 10.1016/j.tripleo.2007.10.003.*

7. Dittmer S., Dittmer M.P., Kohorst P., Jendras M., Borchers L., Stiesch M. Effect of implant–abutment connection design on load bearing capacity and failure mode of implants. Journal of Prosthodontics 2011; 20: 510-516. *doi: 10.1111/j.1532-849X.2011.00758.x*

8. Djebbar N., Serier B., Bachir Bouiadjra B., Benbarek S., Drai A. Analysis of the effect of load direction on the stress distribution in dental implant. Materials & Design 2010; 31: 2097-2101. *doi: 10.1016/j.matdes.2009.10.042*

9. Ekici B. Numerical Analysis of a dental system in three-dimension. Advances in Engineering Software 2002; 33: 109-113. *doi: 10.1016/S0965-9978(01)00056-4*

10. Figueiredo R.B., Barbosa E. R. C., Zhao X., Yang X., Liu X., Cetlin P.R., Langdon T.G. Improving the fatigue behavior of dental implants through processing commercial purity titanium by equal-channel angular pressing. Materials Science and Engineering: A 2014; 619: 312-318. *doi: 10.1016/j.msea.2014.09.099*

11. Geringer A., Diebels S., Nothdurft F.P. Influence of super structure geometry on the mechanical behavior of zirconia implant abutments: a finite element analysis. Biomedizinische Technik. Biomedical engineering 2014; 59(6):501-6. *doi: 10.1515/bmt-2013-0088.*

12. ISO 14801: Dentistry – Implants – Dynamic fatigue test for endosseous dental implants, 2007.

13. Kaman M. O., Celik N., Kilic F. Effects of the dimensions of threads on stress distributions of the dental implants. Turkish Journal of Science and Technology. 2012; 7(2):, 153-166.

14. Karl M., Kelly R. Influence of loading frequency on implant failure under cyclic fatigue conditions, Dental Materials 2009; 25: 1426-1432. *doi: 10.1016/j.dental.2009.06.015*

15. Kayabasi O., Yuzbasioglu E., Erzincanli F. Static, dynamic and fatigue behaviors of dental implant using finite element method. Advances in Engineering Software 2006; 3: 649-658. *doi:10.1016/j.advengsoft.2006.02.004*

16. Kitagawa, T., Tanimoto, Y., Odaki, M., Nemoto, K., Aida, M. Influence of implant/abutment joint designs on abutment screw loosening in a dental implant system. Journal of Biomedical Materials Research Part B-Applied. Biomaterials 2005; 75, 457–463. *doi: 10.1002/jbm.b.30328*

17. Kong L., Gu Z., Hu K., Zhou H., Liu Y., Liu B. Optimization of the implant diameter and length in type B/2 bone for improved biomechanical properties: A three-dimensional finite element analysis, Advances in Engineering Software. 2009; 40: 935-940. *doi: 10.1016/j.advengsoft.2008.12.010*

18. Li T., Hub K., Cheng L., Ding Y., Ding Y., Shao J., Kong L. 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: 446-456. *doi: 10.1016/j.apm.2010.07.008*

19. Lin D., Li Q., Li W., Swain M. Bone remodeling ınduced by dental ımplants of functionally graded materials. Journal of Biomedical Materials Research Part B: Applied Biomaterials 2010; 92: 430-438. *doi: 10.1002/jbm.b.31531*

20. Meriç G., Erkmen E., Kurt A., Tunç Y., Eser A., Influence of prosthesis type and material on the stress distribution in bone around implants: A 3-dimensional finite element analysis. Journal of Dental Sciences2011; 5(3):156−165. *doi: 10.1016/j.jds.2011.02.005*

21. Moraes SL, Pellizzer EP, Verri FR, Santiago JF Jr, Silva JV. Three-dimensional finite element analysis of stress distribution in retention screws of different crown-implant ratios. Computer Methods in Biomechanics and Biomedical Engineering. 2015; 18: 689-696. *doi: 10.1080/10255842.2013.820719*

22. Park S., Won S.Y., Bae T.S., Song K.Y., Park C.W., Eom T.G. ve Jeong C.M. Fatigue characteristics of five types of implant-abutment joint designs. Metals and Materials International 2008; 14: 133-138. *doi: 10.3365/met.mat.2008.04.133*

23. Prados-Privado M., Prados-Frutos J.C., Manchón Á., Rojo R., Felice P., Bea J.A.Dental implants fatigue as a possible failure of implantologic treatment: the importance of randomness in fatigue behaviour. BioMed Research International 2015; 825402. *doi: 10.1155/2015/825402*

24. Schiefer H., Bram M., Buchkremer H.P., Stover D. Mechanical examinations on dental implants with porous titanium coating. Journal of Materials Science: Materials in Medicine 2009; 20: 1763–1770. *doi: 10.1007/s10856-009-3733-1*

25. Sevilla P., Sandino C., Arciniegas M., Martinez-Gomis J., Peraire M., Gil F.J.. Evaluting mechanical properties and degradation of YTZP dental implants, Materials Science and Engineering C: Materials for Biological Applications. 2010; 30: 14-19. *doi: 10.1016/j.msec.2009.08.002*

26. Silva N.R.F.A., Coelho P.G., Fernandes C.A.O., Navarro J.M., Dias R.A., Thompson V.P.. Reliability of one-piece ceramic implant. Journal of Biomedical Materials Research Part B: Applied Biomaterials. 2008; 419-426. *doi: 10.1002/jbm.b.31113*

27. Şahin S., Çehreli M. C., Yalçın E., The influence of functional forces on the biomechanics of implant-supported prostheses-A review. Journal of Dentistry 2002; 20: 271−282. *doi: 10.1016/S0300-5712(02)00065-9*

28. Topkaya H. Dental implant uygulamalarında yorulma davranışının sayısal olarak incelenmesi", Fırat Üniversitesi, Fen Bilimleri Enstitüsü, Yüksek Lisans Tezi, 2014.

29. Yang J., Xiang H.J. A three-dimensional finite element study on the biomechanical behavior of an FGBM dental implant in surrounding bone. Journal of Biomechanics. 2007; 40: 2377–2385. *doi: 10.1016/j.jbiomech.2006.11.019.*