Original Article

Impact of implant diameter and length on stress distribution in osseointegrated implants: A 3D FEA study

R. Eazhil, Siva Vadivel Swaminathan[1], Madhan Gunaseelan[2], G. Vijay Kannan[3], Chandrapandian Alagesan[4]

Departments of Prosthodontics, Chettinad Dental College and Research Institute, Kelambakkam, [3]Asan Memorial Dental College and Hospital, Keerapakkam, Tamil Nadu, India, [1]Ibn Sina National College for Medical Studies, Jeddah, [2]Ram Dental Care, Yanbu, KSA, [4]Future Dental Center, Doha, Qatar

Corresponding author (email: <eazhil235@gmail.com>)
Dr. R. Eazhil, Department of Prosthodontics, Chettinad Dental College and Research Institute, Kelambakkam - 603 103, Tamil Nadu, India.

Received: 26-08-2016 Accepted: 12-10-16 Published: 12-12-16

Abstract

Aims and Objectives: Dimension of dental implant is an important parameter which has a considerable impact on the biomechanical load transfer characters and its prognosis. Excessive stress in the bone–implant interface may result in the failure of the implant. The aim of this study was to evaluate the impact of implant diameter and length on neighboring tissues around the implant. The results of the study will help in developing a scientific methodology to select appropriate implant diameter and length. **Materials and Methods:** In this study, tapered implants of different diameter and length were numerically analyzed using bone–implant models developed from computed tomography generated images of mandible with osseointegrated implants. The impact of various diameters on stress distribution was examined using implants with a length of 13 mm and diameters of 3.5 mm, 4.3 mm and 5.0 mm. Implants with a diameter of 4.3 mm and lengths of 10 mm, 13 mm, 16 mm was developed to examine the impact of various implant length. All materials were assumed to be linearly elastic and isotropic. Masticatory load was applied in a natural direction, oblique to the occlusal plane. The Statistical Package for the Social Sciences software package was used for statistical analysis. **Results:** Maximum von Mises stresses were located around the implant neck. It was demonstrated that there was statistically significant decrease in von Mises stress as the implant diameter increased. **Conclusion:** Within the limitations of this study there was statistically significant decrease in von Mises stress as the implant diameter increased.

Key words: *Dental implant, FEA study, implant diameter, implant length, masticatory load, stress analysis, von Mises stress*

INTRODUCTION

The ability of a successfully osseointegrated implant placed in function to resist bone loss depends on the biomechanical environment. Dental implant functions by translating load to neighboring biologic tissues. Thus, the chief functional design goal of an implant

Access this article online	
Quick Response Code:	Website: www.jispcd.org
	DOI: 10.4103/2231-0762.195518

How to cite this article: Eazhil R, Swaminathan SV, Gunaseelan M, Kannan GV, Alagesan C. Impact of implant diameter and length on stress distribution in osseointegrated implants: A 3D FEA study. J Int Soc Prevent Communit Dent 2016;6:590-6.

is to dissipate and distribute biomechanical loads. The biomechanical load management is dependent on the nature of the applied force and the functional surface area over which the load is dissipated. The principal elements that influence the load transfer at the bone implant interface includes implant geometry, the type of loading, implant material properties, quality and quantity of the surrounding bone, and the nature of the bone–implant interface.[1,2] Important factors in the design of implant that affect load transfer characters include implant diameter and length.[1-5]

Overloading of periimplant bone can be induced by a shortfall in the load transfer mechanisms, mainly due to improper occlusion, prosthesis and/or implant design, and surgical placement. As a sequel, high stress concentrations at the bone-implant interface may arise according to well-supported hypotheses,[6] related strain fields in bone tissue may stimulate biological bone resorption, compromising implant effectiveness.

Clinical research studies have revealed that, in the period of first year after implant loading, the marginal bone loss around the implant ranges from approximately 1.5 mm to 2 mm.[7] This data is in concordance with other three-dimensional (3D) finite element analysis (FEA) studies of dental implants, which indicate that maximum stress occurred around the implant.[4,8,9] Previous studies[8,9] have established that the dimensions of an implant have a definite impact on the stress distribution around the implant. The aim of this study was to evaluate the impact of implant diameter and length on stress distribution in the cortical bone around the implant. To achieve this, models of single tapered threaded dental implants of various diameters and lengths placed vertically into the molar region of the mandible was developed using 3D graphics. The stress distribution around the implants after loading with average masticatory force was calculated by FEA. The purpose of the study is to develop a logical process for selecting the appropriate implant diameter and length in relation to the available bone volume in the patient.

MATERIAL AND METHODS

The purpose of this study is to reproduce the previous studies and analyze the impact of implant diameter and length on stress distribution. To accomplish this several solid models of tapered threaded dental implants (Replace Select Tapered, Nobel Biocare AB, Goteborg, Sweden) with diameters of 3.5–5.0 mm and lengths of 13.0–16.0 mm were developed and studied. For analyzing the impact of different diameters, models with a length of 13 mm and diameters of 3.5 mm, 4.3 mm, and 5.0 mm were developed. Implants with a diameter of 4.3 mm and lengths of 10 mm, 13 mm, and 16 mm were generated to analyze the impact of different implant length. The modeled implant dimensions were on the basis of implant dimensions most frequently used in practice. Three-dimensional solid models of implants and abutments were developed by using a comparative technique which involved high-resolution pictures made with Optical comparator (Deltronics Corp, USA) and actual implants. All 3D solid models (bone segment and implants) were built using CATIA modeling software CATIA V5 R15 (Dassault System Inc., USA). The finite element software ANSYS v 10.0 (South of Pittsburgh, U. S. A.) was used to merge the bone–implant model, as well as to generate and solve the discrete finite element meshes. To develop 3D models of the mandibular molar segment, CT images of human mandible were used. The implant was placed in a vertical direction in the molar region of the model. In this study, the bone segment was simplified to a prism having a quadrangular base and walls of an irregular octagon [Figure 1].[5] The length of the bone segment in the mesio-distal direction was 20 mm to localize the stress around the implant.[10] In the buccal-lingual direction it was 12.5 mm thick and the height was 22.5 mm. The entire volume of the bone was considered to be a homogeneous, isotropic material with the character of cortical bone. Cancellous bone changes its structure after osseointegration and the bone–implant interface becomes more similar to the cortical bone.[10] [Elastic modulus (E) = 1.37 × 10^4 M Pa, Poisson's ratio (u) = 0.3].

Implants were estimated to be completely osseointegrated and were placed at the midspan of the

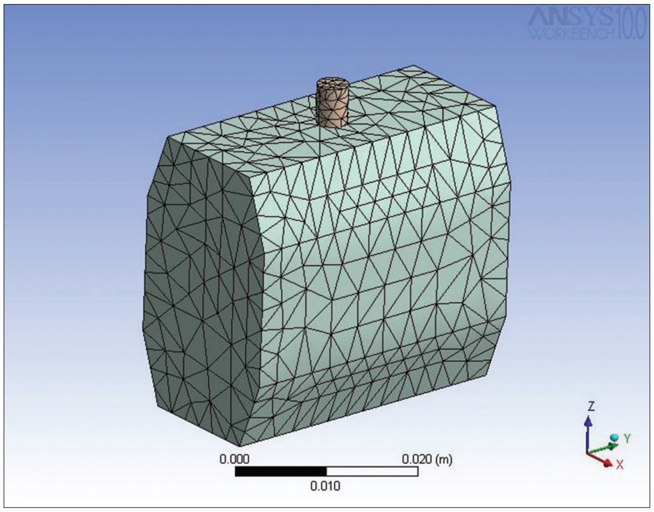

Figure 1: Dental implant and bone model after 'Meshing'

bone segment [Figure 2].[5,10] The developed models consisted of abutment with a height of 4.5 mm. The load was exerted in a 3D nonaxisymmetric pattern on to the centre of the upper surface of the abutment at a height of 4.5 mm from the upper margin of the bone model [Figures 1 and 2]. The implant models were divided into two groups, i.e., Group A and Group B. Implants of Group A investigated the impact of diameter as a factor, implant models with a length of 13 mm and diameters of 3.5 mm, 4.3 mm, and 5 mm were used. Implants of Group B investigated the impact of length as a factor, implant models with diameter of 4.3 mm and lengths of 10 mm, 13 mm, and 16 mm were used. The entire material content was hypothesized to be homogenous and linearly elastic and isotropic.[5,10] The implants and abutments was estimated to be constructed from titanium alloy Ti6Al4V [Elastic modulus (E) = 1.1 × 10^5 M Pa, Poisson's ratio (u) = 0.32]. To simulate the natural average masticatory force in a natural oblique direction of 118.2 N at an angle of approximately 75° to the occlusal plane, 3D loading of the implants was accomplished with force of 114.6 N, 17.1 N and 23.4 N in axial, lingual and distomesial direction respectively [Figure 3].[3,5,10] The magnitude of the force to be applied and its acting point were selected on the basis of previous studies.[10] The ten-node tetrahedral type of element was selected. The element size was 1 mm. The models consisted of 11000 to 14000 elements and 17000 to 22000 nodes depending on the implant size. The models were numerically calculated to evaluate the von Mises stress distributions at the implant bone interface using the FEA software. The values for the loaded elements in the implant model was recorded and averaged, so that results were not significantly affected by numerical errors and mesh asymmetry. In agreement with studies done previously,[2,10] the von Mises stress (σVM) was used as an indicator of the average stress level at the bone–implant interface region. It is considered as a universal measure of load transfer mechanism and the risk for bone failure.

RESULTS

The FE analysis of the implant models showed irregular distribution of stress in the bone around the implants. The maximum stress was concentrated around the neck of the implant on the mesio-lingual region of the socket [Figure 4-7]. This finding was seen in all implant diameters and length was analyzed. The areas with maximum stress for implants of Group A and Group B were calculated [Table 1-4]. The SPSS software package was used for statistical analysis (SPSS for Windows 8.0, SPSS Software Corp., Munich,

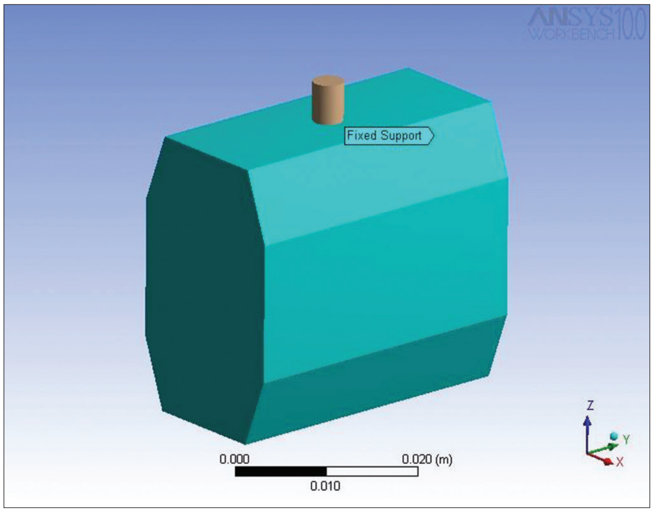

Figure 2: Dental implant and bone model with 'fixed contact' option in the software

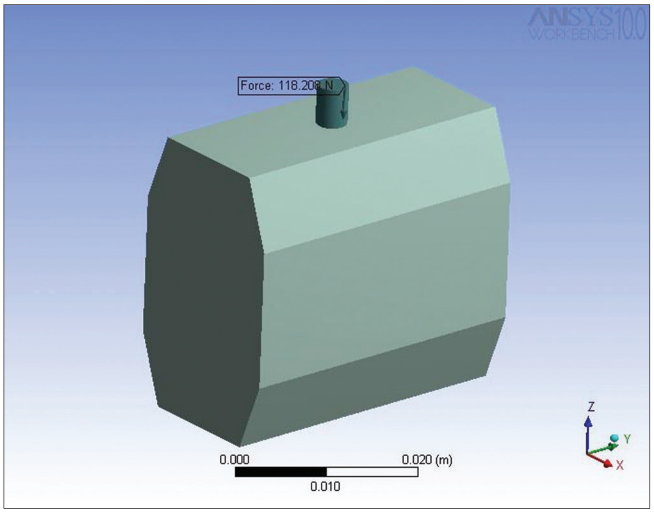

Figure 3: Force of magnitude 118.2 N applied at an angle of 75° to the occlusal plane

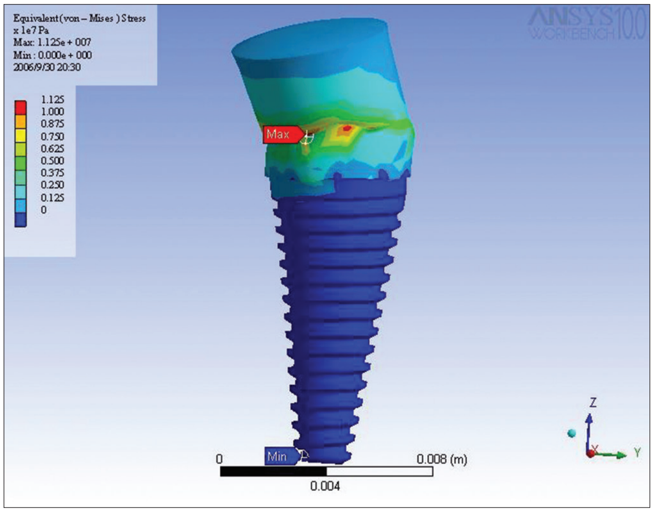

Figure 4: Maximum value of von Mises stress calculated in implant No. 3 of Group A (Diameter – 5.0 mm, Length – 13 mm)

Table 1: Calculation of VMS in each implant of Group A according to masticatory force (118.2 N)

Group A	Diameter (mm)	Length (mm)	Von Mises Stress (VMS) (MPa)
1	3.5	13	16.65
2	4.3	13	13.52
3	5.0	13	11.25

Table 2: Calculation of VMS in bone surrounding each implant of Group A according to masticatory force (118.2 N)

Group A	Diameter (mm)	Length (mm)	Von Mises Stress (VMS) (MPa)
1	3.5	13	3.71
2	4.3	13	3.01
3	5.0	13	2.50

Table 3: Calculation of VMS in each implant of Group B according to masticatory force (118.2 N)

Group B	Diameter (mm)	Length (mm)	Von Mises Stress (VMS) (MPa)
4	4.3	10	17.53
5	4.3	13	13.52
6	4.3	16	12.62

Table 4: Calculation of VMS in bone surrounding each implant of Group B according to masticatory force (118.2 N)

Group B	Diameter (mm)	Length (mm)	Von Misses Stress (VMS) (MPa)
4	4.3	10	3.90
5	4.3	13	3.01
6	4.3	16	2.80

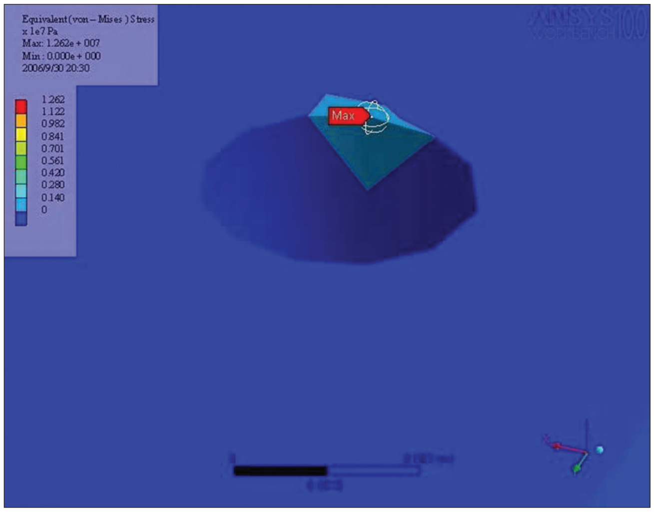

Figure 5: Maximum value of von Mises stress calculated in the bone surrounding implant No. 6 of Group B (Diameter – 4.3 mm, Length – 16 mm)

Germany). Karl Pearson's Coefficient of correlation was calculated to find the significance of implant length and diameter. In the present study, $P < 0.05$ was considered as the level of significance. Table 5 shows Karl Pearson's coefficient of correlation for the significance of the implants in Group A. Table 6 shows Karl Pearson's coefficient of correlation for the significance of the implants in Group B. Implants of the same length but different diameters (Group A) showed significant reduction in stress with increase in diameter which was statistically significant. Implants with same diameter and different lengths (Group B) also showed reduction in stress with increase in length, but it was not statistically significant.

DISCUSSION

Implant biomechanics is an important factor in the long-term survival of dental implants. The ability of implant to transfer physiologic load within a range of 100 to 3000 μ strains to the surrounding bone depends on implant biomechanics.[6] Among the factors that influence implant biomechanics, implant length and implant diameter are the two most common implant design variables of clinical interest under the control of the clinician. This study analyzed the effect of implant diameter and implant length by dividing into two groups of FE simulations. A thorough knowledge about average forces and strain levels generated in function around the implants of various dimensions is a basic requirement to select implants for clinical situations. However, the biomechanical aspects are difficult to analyze by clinical observation/experimental avenues with minimal information and sample variations. FEA has been extensively used in implant dentistry to

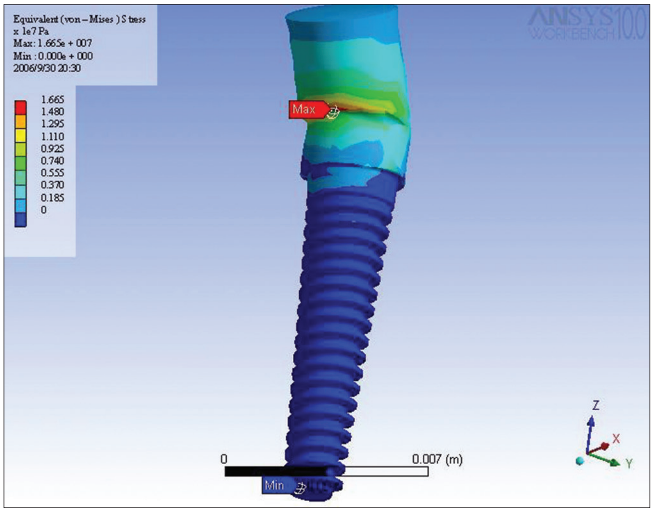

Figure 6: Maximum value of von Mises stress calculated in implant No. 1 of Group A (Diameter – 3.5 mm, Length – 13 mm)

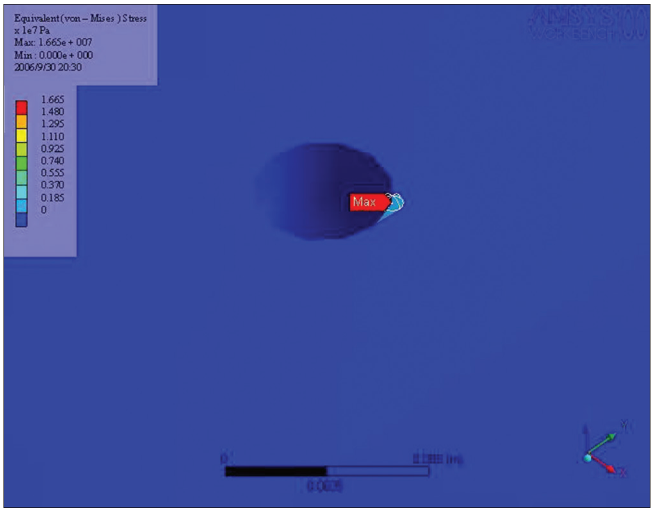

Figure 7: Maximum value of von Mises stress calculated in the bone surrounding Implant No. 1 of Group A (Diameter – 3.5 mm, Length – 13 mm)

Table 5: Correlation of VMS values of implants in Group A

	Mean	Standard deviation	Significance value (*P*)
VMS in implant	13.81	2.71	0.041*
VMS in bone	3.07	0.61	0.040*

**Significant* ($P<0.05$)

Table 6: Correlation of VMS values of implants in Group B

	Mean	Standard deviation	Significance value (*P*)
VMS in implant	14.56	2.61	0.142 (NS)
VMS in bone	3.24	0.58	0.137 (NS)

analyze the biomechanical load transfer at the bone–implant interface. It is possible to accurately reproduce a 3D model of an implant in function and calculate the strain in implant and the surrounding bone. However, the accuracy of the results depend on the accurate reproduction of the 3D model and other factors such as material property, boundary conditions, interface definition, and overall approach to the model.[11]

The FEA study models assumed a state of optimal osseointegration for bone implant interface, i.e., that the cortical bone is absolutely bonded to the implant, which does not occur exactly in clinical situations. However, Papavasiliou concluded that the degree of osseointegration does not affect the stress levels or distributions for axial or oblique loads in finite element study.[12] According to Sato *et al*.[13] in a 3D finite element study, variation in bone stress around an implant were negligible if the length of bone between the modeled implant and the bone segment end was at least 4.2 mm. Therefore, the length of the modeled bone segment was 12.5 mm. In accordance with results obtained by previous studies,[2,5,14] this study confirms that an increase in implant diameter and length reduces the stress magnitudes within cortical bone. The results of this study confirm the conclusion of various studies[2,5,14,15] that implant diameter is an effective design parameter than implant length. In this study, Von Mises stress distributions at the implant–bone interface was localized in the cortical bone around the implant neck in all the models similar to findings of studies done previously.[4,8,9] Carter's hypotheses[6] regarding the effect of the strain level of the bone by hypertrophic responses or bone resorption cannot be computed directly, however, this study established that the risk of bone overload essentially affects regions around the implant neck. Petrie *et al*.[3] concluded that strain near crestal bone area reduces almost by 300% due to increase in diameter as compared to the 165% reduction due to increase in the length of implant.

Effect of diameter

When the diameter of implant is increased, the contact area between the implant and the bone is increased, and therefore, it increases the stability of the implant.[16] Increase in implant strength and fracture resistance can be accomplished by increasing the implant diameter.[17] Crestal bone loss around implants is attributed to occlusal load and it is believed that wider diameter implants reduce the stress around the crestal bone and potential bone loss. Studies have concluded that larger diameter implant is not always the best choice for minimizing cortical bone–implant interface stress and an optimum implant diameter exists for specific patient.[18] The optimum diameter usually correlates with being the largest implant diameter, within morphological limits, causing the least stress when loaded within the surrounding cortical bone and also causing minimal trabecular bone stress.

Effect of length

In implant dentistry, it has been a dictum that longer implants guarantee better prognosis and success. Various FEA studies[2,5,10] have indicated a tendency toward stress reduction on the implant when the length was increased. According to Chun *et al*.,[19] implant length did not play a significant role in reducing the maximum effective stress generated in the surrounding jaw bone when it was beyond a certain length. Guan *et al*.[20] found that, an increase in length reduces the stress, within both cancellous and cortical bone for a wider range of parameters when compared to increasing the diameter. Thakur *et al*.[21] concluded that for reduction of stress intensity at the bone implant interface the implant length should be maximum.

The results obtained by the simulated FE models can be very near to that of the clinical studies despite the simplifications made by assuming the bone to be homogeneous and linearly elastic, fully osseointegrated, and static loading of the implant. The implant bone model was a simulation of the clinical situation. The material properties were simplified to reduce the modeling time without affecting the aim of this study, which is to evaluate the impact of implant diameter and length on stress distribution in the cortical bone around the implant. The results obtained despite the limitations in the modeling procedure gave only a general understanding into the predisposition of stress/stains variations under average conditions, without simulating individual clinical situation. Based on the limited results of this study, it can be concluded that implant diameter and length is an important parameter for biomechanical stress distribution and that maximum stresses occur in the implant neck region. The clinician should select appropriate diameter and length of implant within morphologic limits of the patient after comprehensive clinical and radiographic analysis.

The presence interface resistance limit between bone and implant which is not taken into account in the present FE model can be a compelling design variation to be included in future FE models. For this, the contact and fracture at the implant bone boundary layer should be considered nonlinearly. The simulation of FE models of the entire mandible along with the muscles, the temporomandibular joint, and the stomatognathic system will lead the models to simulate actual clinical situations precisely. Studies have indicated that muscular force action at the bone surface when modeled generated stresses similar to those around the implant. This underlies the significance of modeling the entire mandible and will provide superior qualitative results and insights for future studies.

CONCLUSION

Within the limitations of this study, the 3D FEA study revealed that implant diameter and length affect the mechanisms of biomechanical load translation to the neighboring tissues. Von Mises stress was concentrated in the implant neck region of the bone–implant model. Implant diameter is an effective design parameter than implant length. The increase in the diameter of the implant dissipated the von mises stress better when compared to the increase in the implant length. An optimum implant diameter should be used based on the morphologic limitations of the recipient site.

Financial support and sponsorship

Nil.

Conflicts of interest

There are no conflicts of interest.

REFERENCES

1. Arsalanloo Z, Telchi R, Osgouie KG. Optimum Selection of the Dental Implants According to Length and Diameter Parameters by FE Method in the Anterior Position. Int J Biosci Biochem Bioinformatics 2014;4:265-9.
2. Baggi L, Cappelloni I, Di Girolamo M, Maceri F, Vairo G. The influence of implant diameter and length on stress distribution of osseointegrated implants related to crestal bone geometry: A three-dimensional finite element analysis. J Prosthet Dent 2008;100:422-31.
3. Petrie CS, Williams JL. Comparative evaluation of implant designs: Influence of diameter, length and taper on strains in the alveolar crest. A three-dimensional finite-element analysis. Clin Oral Implants Res 2005;16:486-94.
4. Anitua E, Tapia R, Luzuriaga F, Orive G. Influence of implant length, diameter, and geometry on stress distribution: A finite element analysis. Int J Period Rest Dent 2010;30:89-95.
5. Himmlova L, Dostalova T, Kacovsky A, Konvickova S. Influence of implant length and diameter on stress distribution: A finite element analysis. J Prosthet Dent 2004;9:20-5.
6. Carter DR, Van Der Meulen MC, Beaupré GS. Mechanical factors in bone growth and development. Bone 1996;18(1 Suppl):S5-10.
7. Galindo Moreno P, León-Cano A, Ortega-Oller I, Monje A, O Valle F, Catena A. Marginal bone loss as success criterion in implant dentistry: Beyond 2 mm. Clin Oral Implants Res 2015;26:28-34.
8. Eltas A. Numerical analysis of the effect of implant geometry to stress distributions of three different commercial dental implant system. Cumhuriyet Dent J 2015;18:17-44.
9. Dundar S, Topkaya T, Solmaz MY, Yaman F, Atalay Y, Saybak A, *et al.* Finite element analysis of the stress distributions in peri-implant bone in modified and standard-threaded dental implants. Biotechnol Biotechnol Equip 2016;30:127-33.
10. Demenko V, Linetskiy I, Nesvit K, Hubalkova H, Nesvit V, Shevchenko A. Importance of diameter-to-length ratio in selecting dental implants: A methodological finite element study. Comput Methods Biomech Biomed Engin 2014;17:443-9.
11. Alper Gultekin, Pinar Gultekin and Serdar Yalcin. Finite Element Analysis – New trends and developments 2012 InTech.
12. Papavasiliou G, Kamposiora P, Bayne SC, Felton DA. Three-dimensional finite element analysis of stress-distribution around single tooth implants as a function of bony support, prosthesis type, and loading during function. J Prosthet Dent 1996;76:633-40.
13. Sato Y, Teixeira ER, Tsuga K, Shindoi N. The effectiveness of a new algorithm on a three-dimensional finite element model construction of bone trabeculae in implant biomechanics. J Oral Rehabil 1999;26:640-3.
14. Terzioglu H, Kursunoglu S. Investigation of implant length and diameter variation effect in case of resorption after loading by 3D FEA. Clin Oral Implant Res 2014;25.
15. Yesildal R, Karabudak F, Bayındır F, Zamanlou H, Yıldırım MP, Sagsoz NP *et al.* Effect of Implant Diameter and Length on Stress Distribution for Titanium and Zirconia Implants by Using Finite Element Analysis (FEA). Open Access Library J 2015;2:e1211.

16. Mijiritsky E, Mazor Z, Lorean A, Levin L. Implant Diameter and Length Influence on Survival: Interim Results During the First 2 Years of Function of Implants by a Single Manufacturer. Implant Dent 2013;22:394-8.
17. Lee JH, Frias V, Lee KW. Effect of implant size and shape on implant success rates: A literature review. J Prosthet Dent 2005;94:377-81.
18. Adib MG. Three dimensional finite analysis for different implant diameters. Cairo Dent J 2008;24:131-9.
19. Chun HJ, Cheong SY, Han JH, Heo SJ, Chung JP, Rhyu IC, *et al*. Evaluation of design parameters of osseointegrated dental implants using finite element analysis. J Oral Rehabil 2002;29:565-74.
20. Guan H, Van Staden R, Loo YC, Johnson N, Ivanovski S, Meredith N. Influence of bone and dental implant parameters on stress distribution in the mandible: A finite element study. Int J Oral Maxillofac Implants 2009;24:866-76.
21. Thakur MD, Quazi TZ, Dhatrak PN. Impact of implant pitch and length variation on stress intensity round dental implant-bone interface-by 3d finite element analysis. Int J Engg Sci Res Technol 2016;5.