International Journal of Chemical and Biochemical Sciences
(ISSN 2226-9614)

Journal Home page:www.iscientific.org/Journal.html

© International Scientific Organization

A three-dimensional finite element analysis of impact of implant diameter, length, and thread designs on stress distribution

Thandra Naga Sruthi[1], Srividhya Palicherla[1], Doddy Lokanathan Balaji[2], Naveen Reddy[3], Mohamed Sobhy[3], Azhar Mohammed[4], Madhvika Patidar[5]

[1]Department of Prosthodontics, Government Dental College and Hospital, Kadapa, AP.
[2]Department of Prosthodontics, Priyadarshini Dental College and hospital, Thiruvallur, Tamilnadu.
[3]Department of Prosthodontics, College of Dentistry, Jazan University, Saudi Arabia
[4]Department of Orthodontics and Dentofacial Orthodpedics, Nitte (Deemed to be University), AB Shetty Memorial Institute of Dental Sciences (ABSMIDS), Mangalore, Karnataka, India.
[5]Assistant Professor, Nodal Officer, FICT Center for HIV, Department of Oral Pathology and Microbiology, Government College of Dentistry, Indore, MP, India.

Abstract

Implants are increasingly becoming popular due to their ability to preserve bone quality. There are various implant systems available that must be examined for better outcome. The objective of this research was to study influence of implant length, diameter and thread designs on stress distribution using three-dimensional finite element analysis. Implant systems- Nobel Replace and Nobel Active (Nobel Biocare, Zurich, Switzerland) were analyzed using ANYS software system to quantify the better system of the two. The major and minor stresses and von Misses stress scores were obtained. Statistical analysis failed to demonstrate any superiority of one system over another. Both Noble Replace and Noble Active implant systems were found to have no difference in terms of stress distribution.

Keywords: Tapered, thread design, length, implant, finite element analysis

Full length article **Corresponding Author*, e-mail:vidhyareddy9@gmail.com

1. Introduction

Diameter, length as well as thread design of an implant are parameters that affect distribution of stress to surrounding bony tissues. There are a variety of thread designs available which include- V-shaped, square shaped, buttressed, reverse buttress shaped and spiral thread design [1]. Various factors that affect transfer of load at interface between bone and implant for example, loading types, material properties of an implant as well as prosthetics, geometry of implant, structure of implant surface, quality of design of implant (diameter as well as length), quantity of bone surrounding an implant along with nature of interface between bone and implant [2]. Finite Element Analysis is a method used for mechanical analysis of implants. Finite element analysis has high degree of accuracy and reliability without any involved risk or expense involved with dental implants. This method may be used to study any complex and mechanical issues by means of simulation as well as analysis of distribution of stress. This is achieved by division of three-dimensional problem based geometry in an array comprised of smaller elements. The data comprising of images is obtained with help of computed tomographic scan, 3-D image scanner or by use of magnetic resonance. The outcome of any dental implant can get influenced by large numbers of bio-mechanical factors which may include type of implant loading, material properties of an implant as well as prosthesis, geometry of an implant, structure of an implant surface, nature of implant, quality as well as quantity of surrounding bone, and bone interface and type of surgical procedure [3]. While considering shape of an implant, the main parameters influencing design of an implant which affect mechanisms of transfer of load include- diameter of an implant and length of interface between implant and bone [4], also, pitch of thread, shape as well as depth in threaded implants play an important role [5,6]. When one considers an increase in surface of an implant for better osseous integration, threaded types of implants are usually preferred over smooth surfaced cylindrical implants [7].

Petrie C S et al (2005)compared the effects of implants' diameter, length along with tapering on calculated strains on crestal bone. They concluded that the primary objective in placing any implant is minimization of peri-implant associated strain in alveolar crestal bone. Hence, wide as well as comparatively longer and non-tapered type of implant must be the choice for selecting implants [8]. Lehmann R B et al (2007)stated that conical shaped implant with cantilever shape is highly indicated when compared to a cylindrical shaped implant [9].Images (MRI) for generating Finite Element Analysis model and associated mesh which is required for image analysis. The resultant models contain elements, nodes and pre-defined boundary conditions. The displacement as well as stresses that may be caused by functional loading may be calculated using computerized packages at each of the node. Cylindrical shape of an implant was assessed in this particular study [1].The procedure involving modeling as well as loading on an inserted implant has been considered as causing no harm to underlying bone if resultant forces acting over interface of bone are limited within range of elastic deformation which creates nil or small numbers of micro-cracks. These applied forces have to be within limited range which a surgeon must be able to apply while placing an implant.

Thus, this study was designed with an aim to analyze effects of designs, lengths as well as thread designs of implant by making use of finite element model analysis. Objectives of the study were: 1) To study Major principal stresses; 2) To identify minimum principal stresses and 3) To study Van Misses scores.

2. Materials and methods

Study design: The study was done in department of Prosthodontics. Geometric model generated in current study was based on developing a particular implant model which was fixed to a completely edentulous mandible. The geometry of bone was simplified as well as simulated in the form of cylinder consisting of 2 co-axial cylinders. Inner cylinder represented spongy type of bone with a diameter of 15 mm and a height of 20 mm which has filled internal space of an outer cylinder which was represented as a shell with a thickness of 1 mm that represented cortical bone with a diameter measuring 17 mm and a height of 25 mm.

Methodology: Two different types of implant designs with variations in thread designs were used in current study with a wide range of diameters measuring between 4.5 to 6.5 mm while the length ranged between 10.0 to 13.0 mm.

The Nobel Replace (Group A- T_1) and Nobel Active (Group B -T_2) (Nobel Biocare, Zurich, Switzerland) implants were used in this study. Both of these implants have identical length as well as diameter. The 'Nobel Active' implant possesses variable-thread geometry with grooves that are designed for increasing contact between bone and implant body and has tapered body with gradual increase in thread width. On the other hand, Nobel Replace implant possesses tapered body with 'reverse buttress' grooves which are uniform. Its threads are shorter than Nobel Active implant with thread step of 0.71 mm and neck region measuring 1.5 mm.

Each of the studied implant was then subjected to a total of 3 different types of loading conditions: a) forces of tension measured at 50 N (Ga), b) compression forces at 100 Newton (Gb), c) bending forces applied at force of 15 N (Gc). Base of finite element model were fixed within conditions of the boundary. Loading force was applied over top and middle node of each of the assembly of an implant in all of the studied geometric models. Torque was created by use of 2 equal magnitudes of forces which were in opposite directions and were then applied on 2 points which were located opposite over head of implant diameter.

Inclusion and exclusion criteria: Inclusion criteria was implant system with tapered shape and V-shaped thread designs as one implant group and cylindrical shaped with square thread design as second implant group. Exclusion criteria were none as this was an *in vitro* analysis.

Statistical analysis: Statistical analysis was performed using linear statistics. Solid modeling as well as finite model element analysis was conducted using a PC (personal computer) which had Intel Pentium IV, processor of 2.8 GHz and 2.0 GB RAM.

Meshing software which was used had ANSYS version 9.0. Used element in meshing of all of the 3-dimensional models was 8 nodes Brick element (SOLID45) that had 3 degrees of freedom (translations directed globally). Density of the mesh was other parameter of relevance considered. As geometries were curved, improvement in mesh showed usual effect in improvement of results of any discretely chosen model (which increased accuracy of obtained levels of stress in zones of high gradients of stress). Other effect was an increase in numbers of elements and a reduction in sharp angles that were artificially created with process of geometric model substitution (using the mesh). This reduced artificial peaks of various stress by improving actual representation of geometric model.

3. Results and Discussions

Various implant designs have been analyzed and also, utilized for favoring various mechanisms of Osseo-integration. Use of different implant design modifications such as length and thread designs aims towards promotion of process of Osseo-integration accompanied with quicker and healthier quality of bone formed which can confer better stabilization during process of healing. This allows rapid functional loading of the implant.

The main objective in developing various types of implant designs is to subsequently improve clinical performance in edentulous areas which have poor quantity as well as quality of bone with subsequent acceleration process in healing of bone. Thus, allowing for surgical protocols which allow immediate and early implant loading.

A total of 3 runs on each of designed models were performed which simulated all of the loading force conditions in present study. Preference was given for graphic comparison among the models and cases with loading.

Utilizing the ANSYS VERSION 18.1 software, the analysis of stress scores was carried out by tracking the maximum principal stress (max), minimum principal stress (min), and Von Mises stress values (VM). These variables were chosen to help interpret the results because principle stresses, including compressive and tensile stresses, are thought to be important indicators of stress and strain distributions in materials like bone. When the amplitude of principal stresses is greater than or equal to the ultimate compressive stress on bone, there is a higher chance that osseointegration will fail. It has been discovered that Von Mises' equivalent stress values are crucial for interpreting the stresses present within an implant.

a) *Analysis of maximum principal stress values*: On analyzing both study groups with different thread designs, were 1.42 MPa (G_A - T_1), 0.45 MPa (G_B – T_1) and 1.56 MPa (G_C – T_1) and for Group B categories were 1.34 MPa (G_A – T_2), 1.45 MPa (G_B – T_2) and 1.98 MPa (G_C – T_2) (table 1 and graph 1). following observations were made: The maximum principal stress values for Group A categories

b) Analysis of minimal principal stress values: The minimum principal stress values for Group A categories were found to be -0.54 MPa (G_A - T_1), -1.12 MPa (G_B – T_1) and -0.87 MPa (G_C – T_1) and for Group B categories were -1.23 MPa (G_A – T_2), -0.26 MPa (G_B – T_2) and -1.114 MPa (G_C – T_2) (table 2 and graph 2).

c) Analysis of Von Misses stress scores: The Von Misses stress values for Group A categories were observed as 12.15 MPa (G_A - T_1), 16.12 MPa (G_B – T_1) and 12.18 MPa (G_C – T_1) and for Group B categories were 13.23 MPa (G_A – T_2), 14.78 MPa (G_B – T_2) and 21.14 MPa (G_C – T_2) (table 3 and graph 3). No statistically significant differences were observed in any of the stress scores observed.

A major reason that can lead to delayed failures of implants remains to be the inaccurate selection of type of implants and also, lack in understanding various concepts involving the biomechanical factors associated with implant placement. Himmlova´ L et al (2004)reported that areduction in various types of stress which are exerted by an implant over supporting bone was greatest for implants that had larger diameters when compared with sized implants. They found that the effect of length as well as taper of implant is not as effective when compared to diameter of an implant [10].

Finite Element Analysis has been widely accepted as a theoretical tool which can be used for solving various problems related to engineering. This analytical method has multiple advantages when related to other methods when one is considering any complex situations which are characteristic of real clinical situations. Most models pertaining to Finite Element Analysis work under the assumption of existence of an optimal osseo-integration that means that both cortical as well as cancellous types of bones have a perfect biomechanical bonding with an implant. However, this is not a case in real clinical conditions. Although, Himmlova et al (2004) have concluded that degree of osseointegration affects just deflection and not the levels of stress or patterns of distribution of axial and/or oblique loading in Finite Element model (FEM) analysis [9,10].

However, on comparing with natural masticatory process, these simulated loading with various inclinations in relation to axis of the implants might not be sufficient for completely simulating various oblique mastication forces. Thus, these differences may result in imperfect calculations. Various finite element analysis studies have demonstrated that sites of high stress concentration (for example, implant neck) coincide with predictions related to analysis of stress, exact values of stress that influence osseointegration and longevity of a implant is still largely unknown [11].

In an atrophied and edentulous maxilla, usually six implants are inserted at regular distribution in maxillary arch. These implants are then rigidly connected using a bar super-structure which provides good retention, specifically in cases with advanced maxillary bone atrophy in conformance with this, present study also used this design of prosthesis for rehabilitation using implants.

Also, Lin C-L et al (2005)studied influence of implant's length as well as quality of bone up on bio-mechanical aspects involving alveolar bone and dental implant. They concluded that conditions of implant loading are important in affecting various bio-mechanical aspects when compared with quality of bone as well as length of an implant [11].

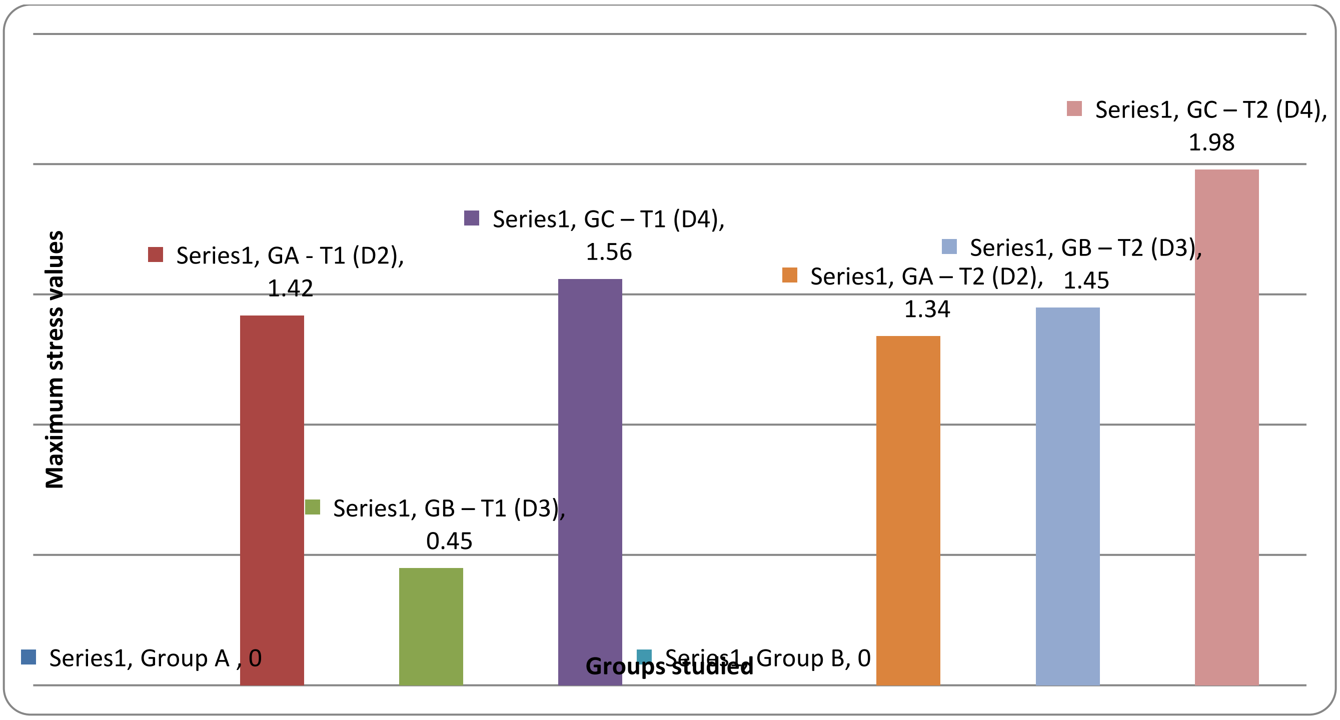
In present study, no statistical difference were noted between both the implant designs, hence, none can be said to be superior over the other. According to Patil et al., the design of the implant's crest module has a significant impact on how much stress is distributed in nearby compact bone [12]. Oswal et al found maximum stresses in cortical bone [13]. The thread design was more considerable in terms of increasing bone stress compared to implant diameter [14].The distribution of stress within the implant body may be impacted by implant design [15].The cortical bone region closest to the implant neck was under the most strain. With bone deterioration and osseointegration rate, the maximum von Mises stress in cortical bone increased, with maximum values [16].

***Limitations associated with Finite element model (FEM) analysis*:**Finite element model analysis is an accurate as well as exact method for analysis of various structures. Although, one must bear in mind that living bodily structures function differently than only objects. Finite Element analysis is dependent up on various types of mathematical calculations that are dependent up on simulating natural biological structures in their natural environment. Living host tissues are not contained within the defined confines of any artificially created parameters as well as scoring systems or values since biology cannot be computed.

Though Finite Element analysis is a perfect theoretical basis for understanding the biological behavior of any studied structure in any environment, it must not be the sole consideration as real time experimental tools accompanied with clinical trials must accompany any finite element analysis for establishing true behavior of any biological system. Also, load application is static in nature whereas in real-life situation during mastication, variations in applied load are present due to variations in muscle force, shape of bone and complex jaw and temporomandibular joint movements. Additionally, variations in thickness of cortical and cancellous bones were not considered while measurement of stress.

Table 1: Table illustrating maximum principal stress

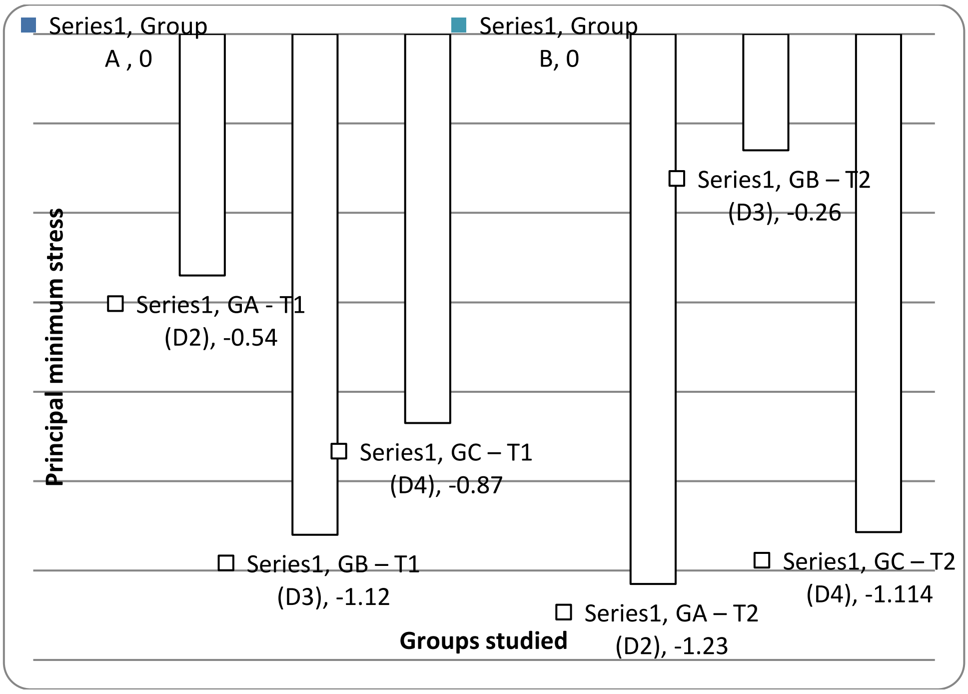
Models	Load (MPa)
Group A	
G_{A} - T_{1}	1.42
G_{B} – T_{1}	0.45
G_{C} – T_{1}	1.56
Group B	
G_{A} – T_{2}	1.34
G_{B} – T_{2}	1.45
G_{C} – T_{2}	1.98



Graph 1: Graph showing maximum principal stress values

Table 2: Table illustrating minimum principal stress

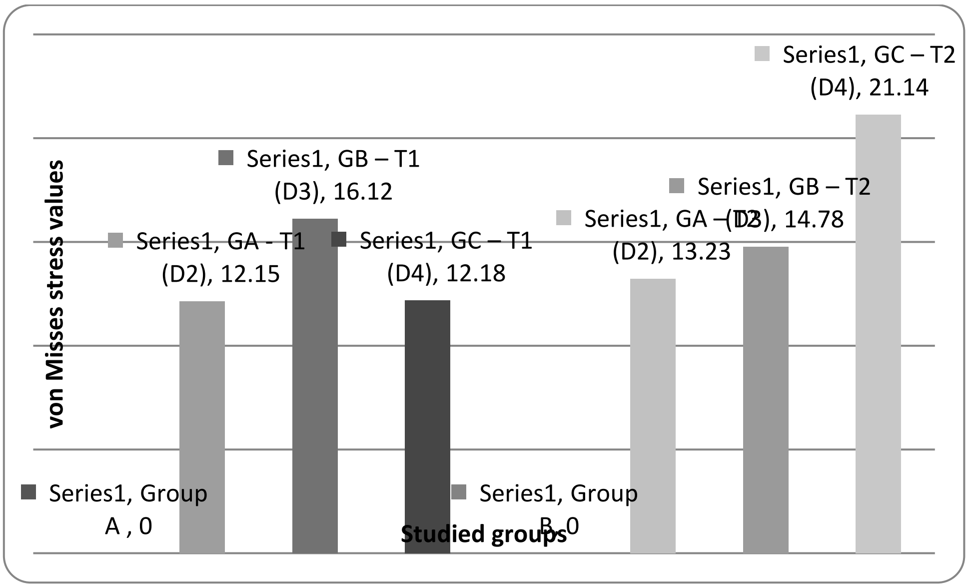
Models	Load (MPa)
Group A	
G_{A} - T_1	-0.54
G_{B} – T_1	-1.12
G_{C} – T_1	-0.87
Group B	
G_{A} – T_2	-1.23
G_{B} – T_2	-0.26
G_{C} – T_2	-1.114



Graph 2: Graph showing minimum principal stress

Table 3: Von Misses stress values

Models	Load (MPa)
Group A	
G_{A} - T_1	12.15
G_{B} – T_1	16.12
G_{C} – T_1	12.18
Group B	
G_{A} – T_2	13.23
G_{B} – T_2	14.78
G_{C} – T_2	21.14



Graph 3: Graph illustrating von Misses stress values

There was a tendency to increase calcium level (–15.64%) compared to the control.Based on the above, it can be concluded that the inclusion of various doses of Cellobacterin-T in the diet does not significantly affect the morphological and biochemical parameters of the blood of experimental chickens.

4. Conclusions

It can be asserted that in present study, all results have been obtained by means of a mathematical model that cannot completely represent complexities associated with a said biological field. Hence, our results may be utilized as early and initial guidelines for performing any other in-vitro stress analysis which must be accompanied with clinical human trials.

References

[1] T.H.Lan, J.K.Du, C.Y. Pan, H.E.Lee, W.H.Chung (2012). Biomechanical analysis of alveolar bonestress around implants with different thread designs and pitches in the mandibular molar area. Clin Oral Investi. 16(2):363-369. doi: 10.1007/s00784-011-0517-z.

[2] M.I.El-Anwar, M.M.El-Zawahary (2011). A three dimensional finite element study on dental implant design. J Genetic Engineering Biotechnol. 9: 77–82. doi:10.1016/j.jgeb.2011.05.007.

[3] F.E.Lambert, H.P.Weber, S.M.Susarla, U.C. Belserand, G.O.Gallucci (2009). Descriptive analysis of implant and prosthodontic survival rates with fixed implant-supported rehabilitations in the edentulous maxilla. J Periodontol. 80(8):1220–1230. doi: 10.1902/jop.2009.090109.

[4] T.Li, K.Hu, L.Cheng (2011). 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.35(1):446–456. DOI10.1016/j.apm.2010.07.008

[5] C.C.Lee, S.C.Lin, M.J. Kang, S.W.Wu, P.Y.Fu (2010). Effects of implant threads on the contact area and stress distribution of marginal bone. J Dent Sci. 5(3):156–165.

[6] H.L.Huang, C.H.Chang, J.T.Hsu, A.M.Fallgatter, C.C.Ko (2007). Comparison of implant body designs and threaded designs of dental implants: a 3-dimensional finite element analysis. Int J Oral and Maxillofacial Implants. 22(4):551–562. PMID: 17929515

[7] N.Sykaras, A.M.Iacopino, V.A.Marker, R.G.Triplett, R.D.Woody (2000). Implant materials, designs, and surface topographies: their effect on osseointegration. A literature review. International J Oral Maxillofac Implants. 15(5):675–690.

[8] C.S.Petrie, J.L.Williams (2005) .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. 16(4):486-494. doi: 10.1111/j.1600-0501.2005.01132.x.

[9] R.B.Lehmann, C.N.Elias (2007). Influence of Cantilever on Stress Distribution In Bone Around Dental Implants: A Three-Dimensional Finite Element Analysis. 19th International Congress of Mechanical Engineering. November 5-9, Brasília. Proceedings of COBEM 2007

[10] L. Himmlova, T.Dosta´lova, A. Ka´covsky, S.Konvicˇkova´ (2004). Influence of implant length and diameter on stress distribution: A finite element analysis. J Prosthet Dent. 91:20-25.doi: 10.1016/j.prosdent.2003.08.008.

[11] C.L.Lin, Y.C.Kuo, T.S.Lin (2005). Effects of dental implant length and bone Quality on biomechanical responses in bone around implants: a 3-d nonlinear finite element analysis. Biomed Eng Appl Basis

Comm. 17: 44-49. https://doi.org/10.4015/S1016237205000081
[12] S.M. Patil, A.S. Deshpande, R.R.Bhalerao, S.B.Metkari, P.M. Patil (2019). A three-dimensional finite element analysis of the influence of varying implant crest module designs on the stress distribution to the bone. Dent Res J (Isfahan). 16(3): 145–152.
[13] M.M.Oswal, U.N.Amasi, M.S.Oswal, A.S.Bhagat (2016). Influence of three different implant thread designs on stress distribution: A three-dimensional finite element analysis. J Indian Prosthodont Soc.16:359-65.
[14] ZF.Zor, Y.,Kilinç, E.Erkmen, A.Kurt (2021). Evaluation of biomechanical effects of different implant thread designs and diameters on all-on-four concept. ARIESDUE.13(3):101-108
[15] B. Houshmand, M.TalebiArdakani, A.Moscowchi, I.Z.Oskoui (2022). Effect of Implant Design on Stress Distribution: A Finite Element Study. Journal of Long-Term Effects of Medical Implants. 32(4): 39-45.
[16] Y.Yang, Y.Liu, X.Yuan, M.Ren, X.Chen, L. Luo, et al (2023). Three-dimensional finite element analysis of stress distribution on short implants with different bone conditions and osseointegration rates. BMC Oral Health. 23 (220):1-14