International Journal of Scientific Engineering and Research (IJSER)
www.ijser.in
ISSN (Online): 2347-3878, Impact Factor (2014): 3.05

Determination of Stresses Induced In the Jawbone after Implant of Tooth Using Finite Element Method

Prof. Vivek C. Pathade[1], Shruti N. Keche[2]

[1]Department of mechanical Engineering, Data Meghe Institute of Engineering Technology & Research, Wardha, India
[2]Department of CAD/CAM Engineering, Data Meghe Institute of Engineering Technology & Research, Wardha, India

Abstract: ***The aim of this study was to evaluate the effect of dental implant with different length and diameter in surrounding bone of implant under normal vertical load. A comparative study between short length and diameter and large length and diameter of implant is done so as to obtain best feasible length and diameter of implant that makes less deformation by using finite element analysis (3D FEA). Material and Methods - The FEM and analysis techniques are used to replicate and evaluate the deformation in surrounding bone and stress profile created the mandible single tooth implant surrounding bone. Homogeneous, linear, elastic material behavior for the jawbone and titanium implant of young's modulus 110Gpa were assumed. Results - Maximum stress areas were located at the implant neck, and possible overloading could occur in compression in compact bone and in tension at the interface between cortical and spongy bone. Stress values and concentration area were decreased for surrounding bone when implant length and diameter increased. For implants with comparable diameter and length, compressive stress values at surrounding bone were reduced when low crestal bone loss was considered. Conclusion - Implant dimension and bone quality affects load transmission mechanisms. Implant length12.5mm and 4mm diameter gives fissile stress distribution and less deformation in surrounding bone.***

Keyword: Implant length and diameter, load, Reduction of bone, stress distribution, FEM

1. Introduction

Dental implants are biocompatible screw-like titanium "fixtures" that are surgically placed into a jawbone to replace missing tooth. When, the natural tooth is lost in cases of accidental injury, damage due to oral infections or deformities at birth. It is fixed in jawbone with the help of surgical procedure and the decision on selection of implant is generally based on judgment and experience of clinician [1]. The process of implantation requires an optimum stress profile in order to maintain a strong and healthy jawbone. The success of a dental implant is the manner in which stresses are transferred to the surrounding bone and biomechanical bonding with surrounding bone of implant; it is also called as Osseo integration phenomenon in dental field. Load transfer from implants to surrounding bone depends on the type of loading, the bone to implant interface, the length and diameter of the implants, the shape and characteristics of the implant surface, the prosthesis type, and the quantity and quality of the surrounding bone of implant[2,3]. The biomechanical factors have been stressed by various authors [4]. Micro movement of the implant fixture and excessive stress at the implant-bone interface has been suggested as potential cause for peri-implant bone loss and failure [5].

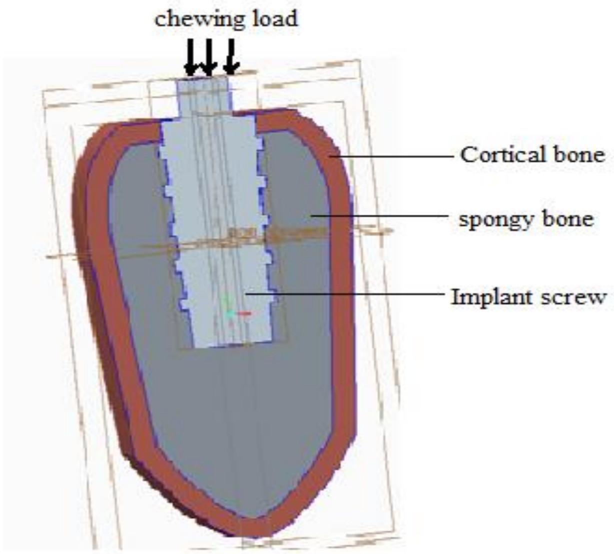
In many cases it has been seen that, after an interval of time this implant gets loose due to the deformation of bone which happen in surrounding bone of implant due to the continuous impact of load on dental implant. The stresses and reduction of bone around an implant in human jawbone are to be determined with increasing diameter and length by using finite element method. CAD and FEM environment provides various advantages of modeling and analysis over other conventional methods. Hence it can be used in dental industry for its data accuracy and data interpretational capability [6]. In this paper took the study of the techniques used by researchers working in this field for analyzing the effects of various parameters on deformation levels and ultimately the success of implantation procedure. A thorough understanding of this phenomenon might lead to a reduction in the surrounding bone of implant within the jawbone [7]. Bone is a self-adaptive material which means that when the surrounding stress is changed, the bone tissue structure is adjusted by itself to suit the new loading environment. This is known as the bone remodeling. Such remodeling includes both internal modification of apparent bone density and external alteration of bone shape. Bone remodeling induced by the change of normal biological stress is one of the most important factors causing implant failure. This is the so-called stress-shielding effect. Using the finite element method and parameterized optimum design technique, the stress-shielding effect can be minimized to a maximum extent by performing multi parameter optimization of implant, there by guaranteeing the success rate of implantation.

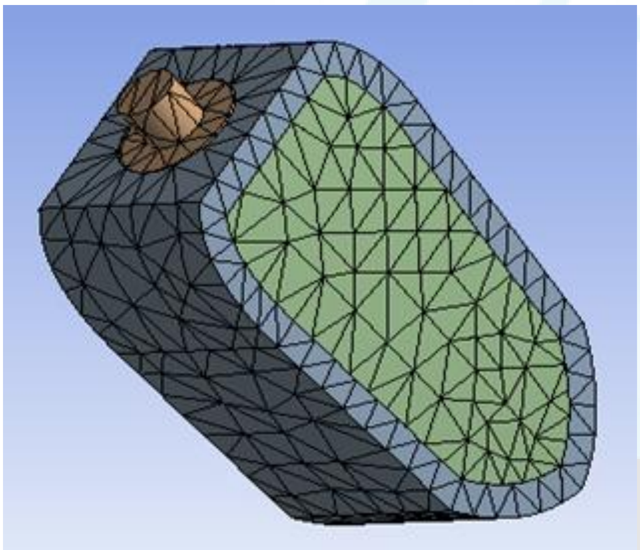
2. Material and Method

The FEA is an accepted theoretical technique used in the solution of engineering problems. It has been proven to be a precise method for evaluating dental implant systems and it offers many advantages over other methods. FEA modeling not only can simulate complex geometric shapes and material properties, but also can simulate various boundary conditions, which are difficult to replicate in experiments.

Baiamonte et al. [8] compared in vitro results of the strain on a loaded monkey mandible that had osteointegrated titanium implants with the results obtained by means of the FEA. Since the results were in excellent agreement it was concluded that FEA is also applicable to dental systems. It involves the development of a mathematical model of a continuous structure divided into a system of discrete components or elements. These components are connected at nodal points where stress and displacements determined.

The accuracy of the 3D method is proportional to the number of nodes and elements in the mathematical model [17, 18]. Homogeneous, linear, elastic material behavior for the jawbone and titanium implant of young‟s modulus "110Gpa" were assumed.





Surrounding jawbone divides in two sub-bones cortical bone and spongy bone. Different length and diameter of implants were considered for fissile result in three different cases of humane mandibular. In the acceptability of these kind of works done by the FEA, many factors have great importance.

3. Loading In FEM

The purpose of this study was to evaluate the effects of the three different dimensions of implant on deformation and stress distribution under 500N vertically normal loading condition in mandibular posterior edentulism with standard-diameter implants using 3D FEA. Load i.e. chewing load was applied on single tooth implant which had 110Gpa young‟s modulus loading condition were same in three cases and the cortical layer has a Poisson‟s ratio of 0.3,0.31 and for spongy bone 0.29 ,0.3 and a modulus of elasticity (Young‟s modulus) were differs with quality of bone. Fixed support given to the surrounding jawbone for static condition. According to the Saint Venant‟s Principle, the actual force system may be replaced by a statically equivalent load system and the distribution of stresses and strains are only affected near the regions of applied loads. The binding conditions of the bone-implant interface and the loading conditions of the model are the major factors [19,9].

The force distribution in the regions of the implants where the loads were applied may not simulate the real distribution referring to Saint Venant‟s principle but the main concern of this study was to compare the stress values in the bone due to the change of the dimensions of implants, it differs from short diameter and length to the large diameter and length of implant. Furthermore, the bone was quite far away from the load boundary condition.

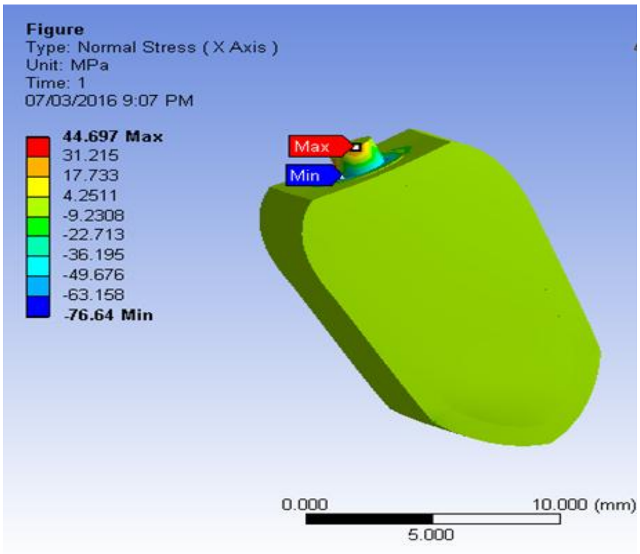
$$e = \frac{P}{A_{sb} \times E}\left(1 - \frac{1}{m}\right)$$

Above equation show the deformation of surrounding bone of implant considering 500N average vertical chewing load impact on implant tooth. Where internal strain in jawbone is denote by e and area of bone considering surrounding bone of implant.

4. Result

Principle stress and Von Mises stress are equally used. This study focused on the stress formed in both cortical and spongy bone around the implants. Von Mises stress values are defined as the beginning of deformation for ductile materials such as dental implants, thus, these values may be important to interpret the stress occurred within the implants. Maximum and minimum principal stress (Pmax and Pmin), offering the possibility of making a distinction between tensile and compressive stress are used in this study. The bone can be classified as brittle in engineering sense therefore the principle stress values are appropriate while analyzing such a work. Regarding the figures stress patterns appeared as contour lines with different colors. The corresponding color scale is represented near the figures and the tensile and compression values are in positive and negative signs respectively to have a conventional engineering representation.

Equivalent von-mises stress differs with dimension and bone quality of surrounding bone of jawbone. It analyzed "133.57Mpa, 314.02Mpa, 204.59Mpa" by considering different cases. Its details are shown in following table. It observed around the implant.



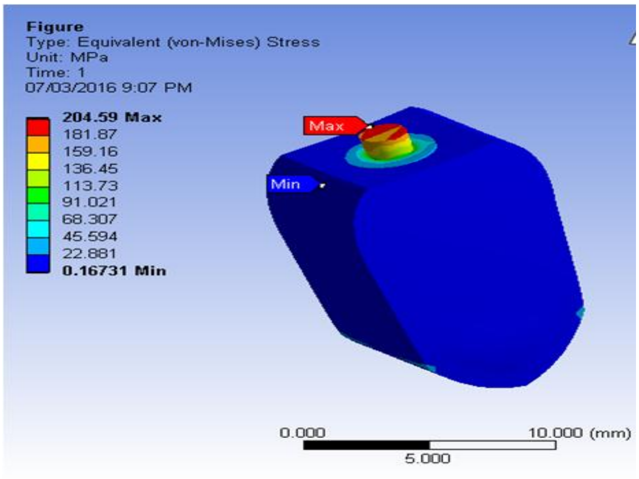
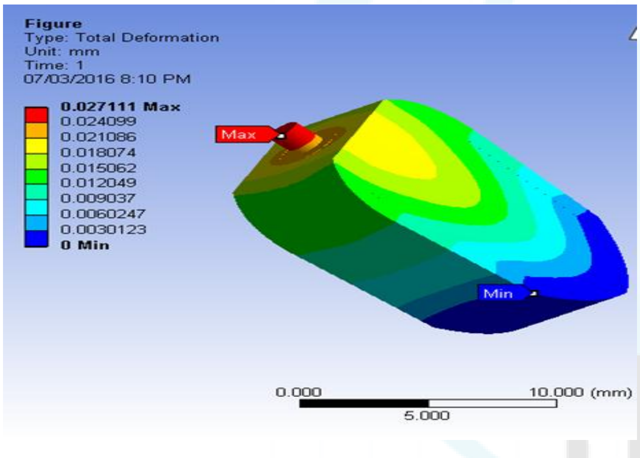
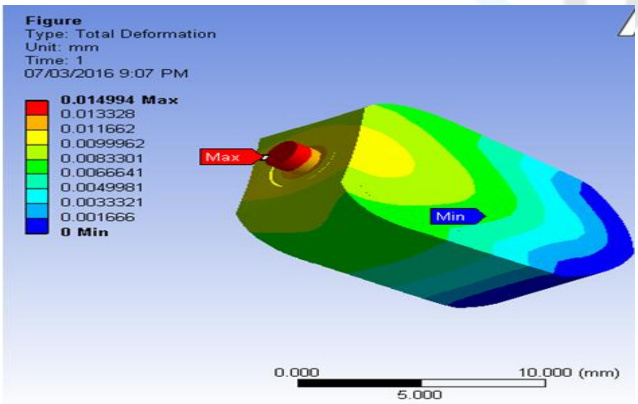


Table 1: Stress and strain values with considering different dimensions of implant screw in different qualities of surrounding jawbone

Implant Dim(mm)	*Young's modulus of implant(GPa)*	*Poison's ratio of implant*	*Young's modulus of cortical bone*	*Young's modulus of Spongy bone*	*Analytical stress (MPa)*	*Analytical strain (MPa)*
10-3.3	110	3.3	12.6	2.1	63.366	0.027
12.5-4	110	3.3	13.7	3.95	26.491	0.0143
13.5-4	110	3.3	13.2	3.1	44.697	0.0149





According to the above table deformation in surrounding bone of implant reduces by increasing length and diameter. Fissile length of implant is "12.5 mm and 4 mm" diameter. Deformation depends on quality of bone of jawbone. In short implant of length "10mm" and diameter "3.3" internal deformation was "0.027".

For short implants, increasing diameter improves its effect on cortical bone (as it reduces generated stresses). In these cases, both side and cross-sectional areas were increased. Increasing diameter reduces the ratio between side and cross sectional areas. Comparing stress values on cortical bone while side area and cross-sectional area increase revealed a reduction in cortical bone stress values by increasing side area.

5. Conclusion

The finite element method (FEM) has been used extensively to predict the biomechanical performance of various dental implant designs as well as the effect of clinical factors on the success of an implantation. It has been identified that the principal difficulty in simulating the mechanical behaviour of a dental implant is the modelling of the living human bone tissue and its response to applied mechanical forces. Research has been conducted on the length, diameter and shape of implants as well as the biomechanical bond formed between the implant and the jawbone. An in-depth understanding of the stress profiles experienced by the implant and more importantly within the surrounding jawbone can be gained through the use of the FEM.

This study focused on the stress distribution and deformation in the surrounding bone due to the influence of implant dimension. Within the limitations of this study, numerical simulations showed that implant design, in terms of both implant diameter and length, crestal bone geometry and placement site affect the mechanisms of load transmission. Stress distribution pattern changes according to the bone quality and increasing diameter and length of implant. Values of stresses may alter a in case of changing implant diameter and/or length. Wide implants

(4 mm diameter) behavior was noticeably affected when its length was increased. On the other hand, small diameter implant behaviors were dramatically enhanced by increasing implant length (side area). Cortical periimplant areas that could be affected by overloading were influenced primarily by implant diameter, irrespective of bone-implant interface length. However, an increase in implant length reduced stress gradients at the spongy periimplant region.

References

[1] Ratnakar R. Ghorpade and Shailesh S. Yelekar "Computational and experimental studies in threaded dental implant research" e-Journal of Dentistry Oct - Dec 2013 Vol 3 Issue 4 457-462
[2] Friberg B, Jemt T, Lekholm U. Early failures in 4,641 consecutively placed Branemark dental implants: a study from stage 1 surgery to the connection of completed prostheses. Int J Oral Maxillofac Implants 1991;6(2):142-146
[3] Albrektsson T, Johansson C. Experimental and clinical studies of different ways to improve the outcome of implants placed in bone of deficient quantity and quality. J Parodontol Implantol Orale 2000;19:271-288.
[4] Stegaroiu R, Sato T, Kusakari H, Miyakawa O. Influence of restoration type on stress distribution in bone around implants: a three dimensional finite element analysis. Int J Oral Maxillofac Implants 1998;13:82–90.
[5] Ciftci Y, Canay S. "The effect of veneering materials on stress distribution in implant-supported fixed prosthetic restorations. Int J Oral Maxillofac Implants"2000;15:571–82.
[6] Holmes DC, Loftus JT. Influence of bone quality on stress distribution for endosseous implants. J Oral Implantol 1997;23:104–11.
[7] Dhiraj Agnihotri1 , Amol More2 , Dr. P.G.Patil3 "A review on application of cad and fem technology in design of taper dental implant"(IJIRSET) Vol. 2, Issue 6, June 2013, 2091-2096
[8] El-Anwar, M.I., El-Zawahry, M.M. and E-Mofty, M., „Load transfer on dental implants and surrounding bones", Australian Journal of Basic and Applied Sciences-2012, Vol.6(3); 551-560.
[9] Baiamonte T, Abbate MF, Pizzarello F, Lozada J, James R. "The experimental verification of the efficacy of finite element modeling to dental implant systems. J Oral Implantol" 1996;22:104–10.
[10] Lara L. Tulla, "Measuring distance lateral to the bone-implant interface with various stages of loading: a histomarphometric analysis in baboon"., University of Florida, December 2003
[11] Prasad Konda and Tarannum S A. "Basic principles of finite element method and its applications in orthodontics" (JPBMS), Vol. 16, Issue 16(11), ISSN NO- 2230 – 7885.
[12] Vidyasagar, L. and Apse, P., "Dental implant design and biological effects on bone-implant interface", Baltic Dental and Maxillofacial Journal-2004, 6(2); 51-54.
[13] Pierrisnard L, Renouard F, Renault P, Barquins M: Influence of Implant length and bicortical anchorage on implant stress distribution. Clin Implant Dental Relat Res.2003; 5(4):254-262.
[14] 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.2005Aug; 16(4):486-494.
[15] M Ángeles Fuster-Torres , Salvador Albalat-Estela , Mariano Alcañiz-Raya , María Peñarrocha-Diago "CAD / CAM dental systems in implant dentistry: Update" Med Oral Patol Oral Cir Bucal. 2009 Mar 1;14 (3):E141-5.
[16] Barıs¸ S¸ims¸eka,Erkan Erkmena, Dervis¸ Yilmaz a, Atılım Eser b "Effects of different inter-implant distances on the stress distribution around endosseous implants in posterior mandible" Medical Engineering & Physics 28 (2006) 199–213
[17] Omer Pektas, "Desidn and mechanical analysis a new dental implant that would minic natural tooth with a periodonatal ligament." Middle east technical university,feb-2012,15-89
[18] Lewinstein, I., Banks-Sills, L., and Eliasi, R., 1995, "A finite element analysis of a new system (IL) for supporting an implant-retained cantilever prosthesis. International Journal of Oral Maxillofacial Implants" 10(3), 355-366.
[19] Mihalko, W.M., May, T.C., Kay, J.F., and Krause, W.R., 1992, Finite element analysis of interface geometry effects on the crestal bone surrounding a dental implant. Journal of Implant Dentistry, 1(3), 212-217.
[20] Nishihara, K., and Nakagiri, S., 1994, Biomechanical studies on newly tailored artificial dental root. Biomedical Material Engineering, 4(3), 141-149.
[21] A. S. Bonnet, M. Postaire, P. Lipinski, „Biomechanical study of mandible bone supporting a four-implant retained bridge. Finite element analysis of the bone anisotropy and foodstuff position", Medical Engineering and Physics-2009, Elsevier.
[22] Y. Morita, L. Quian, M. Todo, Y. Matsushita, K. Arakawa, „Strain distribution around dental implants in Cortical/Cancellous bone models using DIC method", Proceedings of the SEM Annual Conference-2009.
[23] Chang SH, Lin CL, Hsue SS, Lin YS, Huang SR. Biomechanical analysis of the effects of implant diameter and bone quality in short implants placed in the atrophic posterior maxilla. Med Eng Phys 2012; 34:153-160.
[24] Van Staden RC, Guan H, Loo YC. "Application of the finite element method in dental implant research." Computer Methods Biomechanical Biomed Engine 2006;9:257-70