ISSN 2395-1621

Effect of Screw Profile on Stress Distribution Pattern of Dental Implants by FEA

[#1]RaiphaleGovind, [#2]Shinde B.M.

[1]govindaraiphale@gmail.com*1
2bharat.shinde@zealeducation.com

[1](M.E Student ZES's Dnyanganga College of Engineering and Research, Narhe.)
[2](Assistant Professor ZES's Dnyanganga College of Engineering and Research, Narhe.)

ABSTRACT

Dental implants constitute a well-established approach for substitute of lost teeth with titanium being the most preferential material for implantation. However, titanium has its confines in esthetically demanding cases and neither the form nor material of such implants has changed much over the past 40 years. Immediate implantation is used to overcome the disadvantages of conventional implantation which in turn has many disadvantages owing to the incongruence of the implant to the extraction socket. The commonly used method for testing of new implant prosthesis is in vitro which involves direct involvement of human. But use of modern techniques can help one decide the diameter and screw thread profile. In finite element analysis the domain is divided in number of small elements. In finite element analysis one can also simulate different material models and analyze the actual working conditions. In this project work the size of the implant is selected. A 3-D CAD model is modeled with appropriate CAD software. A finite element model is created representing the bone and dental implant. The result obtained from finite element analysis shows the stresses and deformation at the contact locations of bone and implant. By varying the diameters and screw profiles one can come to conclusion of the optimum design of the implant prosthesis.

Keywords— Dental implant, Optimum, FEA.

ARTICLE INFO

Article History

Received :18th November 2015

Received in revised form :

19th November 2015

Accepted : 21st November , 2015

Published online :

22nd November 2015

I. INTRODUCTION

A dental implant as a screw type biomaterial is a functional load transfer structure to substitute for lost or partially damaged teeth. Natural teeth consist of the crown and the root. The crown is the visible section that is covered with white enamel. Supporting the crown is the tooth root which extends into the jawbone. The root is the part of tooth that is effectively replaced by an implant. There are commonly three parts to what is described as an implant - the implant device itself (which is inserted directly into the bone); the abutment - the piece that connects the implant device to the third part - the overlying crown or denture. Today's implants are predominantly made of titanium, a metal that is bio-compatible and offers strength and durability as well as a unique property of fusing directly to bone - the process known as osseointegration. Other materials, such as zirconium, might be used to make implants in the future. But for now, these materials have not been perfected for general use. Dental implants work by a process known as osseointegration, which occurs when bone cells attach themselves directly to the titanium surface, essentially locking the implant into the jaw bone. This process was first discovered by a Swedish researcher, Per-Ingvar Brånemark, in the 1960's. Placing dental implants into the jaw bones by controlled surgical procedures allow them to "osseointegrate."

Osseointegrated implants can then be used to support prosthetic tooth replacements of various designs and functionality, replacing anything from a single missing tooth to a full arch (all teeth in the upper and lower jaw). These replacement teeth are usually made to match the natural enamel colour of each patient which offers a completely natural appearance and a whole new smile.

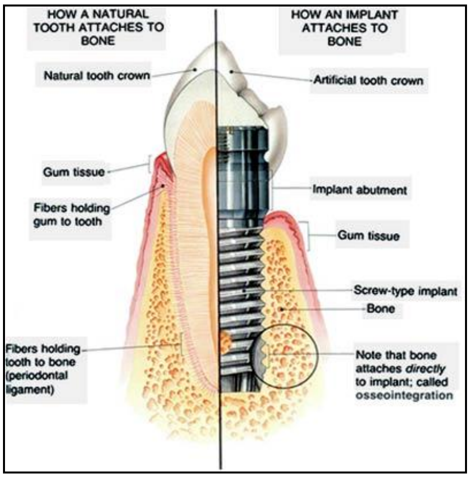


Fig.1 Structure of dental implant

Because implants fuse to your jawbone, they provide stable support for artificial teeth. Dentures and bridges mounted to implants won't slip or shift in your mouth — an especially important benefit when eating and speaking. This secure fit helps the dentures and bridges — as well as individual crowns placed over implants — feel more natural than conventional bridges or dentures. For some people, ordinary bridges and dentures are simply not comfortable or even possible, due to sore spots, poor ridges or gagging. In addition, ordinary bridges must be attached to teeth on either side of the space left by the missing tooth. An advantage of implants is that no adjacent teeth need to be prepared or ground down to hold your new replacement tooth/teeth in place.

II. LITERATURE REVIEW

There are some literatures available on the study of dental implants. The attempt here is to highlight relevant literature evidences.

Lucie Himmlova et.al published a paper Influence of Implant Length and Diameter on Stress Distribution: A Finite Element Analysis. Their study reveals that Implant size influences the area of possible retention in the bone; factors such as occlusion, masticatory force, the number of implants, and implant position within the prosthesis affect the forces acting on the bone adjacent to implants. Paper also states that implant diameter, shape, and load direction influence stress distribution. Alveolar bone reduction caused by extraction wound healing, wearing of removable dentures, and bone injuries, together with anatomic structures (such as the canine fossa, antrum, nasal cavity or mandibular canal), may limit implant size or require implant placement into positions for which angled abutments are needed. This finite element study showed that increased implant diameter better dissipated the simulated masticatory force and decreased the stress around the implant neck. The highest reduction in stress compared to the reference implant (100%, 3.6 mm wide, 12 mm long) was obtained for the diameter of 4.2 mm (a 31.5% decrease). From a biomechanical perspective, the optimum choice was an implant with the maximum possible diameter allowed by the anatomy. In this study, the effect of implant length was less notable [1].

Chun-lilinet.al, published paper Effects of Dental Implant Length and Bone Quality on Biomechanical Responses in Bone around Implants: A 3 D Non-Linear Finite Element Analysis

The aim of this study was to evaluate the influence of implant length and bone quality on the biomechanical aspects in alveolar bone and dental implant using non-linear finite element analysis. Two fixture lengths (8 and 13mm) of Frialit-2 root-form titanium implants were buried in 4 types of bone modeled by varying the elastic modulus for cancellous bone. Contact elements were used to simulate the realistic interface fixation within the implant system. Axial and lateral (buccolingual) loadings were applied at the top of the abutment to simulate the occlusal forces. The simulated results indicated that the maximum strain values of cortical and cancellous bone increased with lower bone density. In addition, the variations of cortical bony strains between 13mm and 8mm long implants were not significantly as a results of the same contact areas between implant fixture and cortical bone were found for different implant lengths. Lateral occlusal forces significantly increased the bone strain values when compared with axial occlusal forces regardless of the implant lengths and bone qualities. Loading conditions were found as the most important factor than bone qualities and implant lengths affecting the biomechanical aspects for alveolar bone and implant systems. The simulated results implied that further understanding of the role of occlusal adjustment influencing the loading directions are needed and might affect the long-term success of an implant system [2]

M. Sevimayet.al Published a paper Three-dimensional finite element analysis of the effect of different bone quality on stress distribution in an implant-supported crown.

The purpose of this study was to investigate the effect of 4 different bone qualities on stress distribution in an implant-supported mandibular crown, using 3-dimensional (3-D) finite element (FE) analysis. Primary implant stability and bone density are variables that are considered essential to achieve predictable osseointegration and long-term clinical survival of implants. Information about the influence of bone quality on stress distribution in an implant-supported crown is limited. Available bone is particularly important in implant dentistry and describes the external architecture or volume of the edentulous area considered for implants. In addition, bone has an internal structure described in terms of quality or density, which reflects the strength of the bone.

The density of available bone in an edentulous site is a determining factor in treatment planning, implant design, surgical approach, healing time, and initial progressive bone loading during prosthetic reconstruction.The influence of the implant design on stress concentration in the bone during loading and indicated that the implant design was a significant factor influencing the stress created in the bone [3]

L. Kong et.al, presented the work Selection of Optimal Dental Implant Diameter and Length in Type Iv Bone: A Three Dimensional Finite Element Analysis

This study aimed to create a 3D finite element model for continuous variation of implant diameter and length, thereby identifying their optimal range in type IV bone under biomechanical consideration. Implant diameter ranged from 3.0 to 5.0 mm, and implant length ranged from 6.0 to 14.0 mm. The results suggest that under axial load, the maximum Von Mises stresses in cortical and cancellous bones decrease by 50% and 27%, respectively; and under buccolingual load, by 52% and 60%, respectively. Under these two loads, the maximum displacements of implant abutment complex decrease by 39% and 43%, respectively. These results indicate that in type IV bone, implant length is more crucial in reducing bone stress and enhancing the stability of implant abutment complex than implant diameter. Biomechanically, implant diameter exceeding 4.0 mm and implant length exceeding 9.0 mm are the combination with optimal properties for a screwed implant in type IV bone [4].

From the literature reviewed it is observed that finite element analysis helps in designing the basic parameters of the dental implant. From the papers the loading conditions were decided to use for this dissertation work [2]. The material properties required for bone is noted down from the paper [4] published by T. Li, L. Kong et al..The use of numerical analysis helps in designing the model of implant was explained by From the literature review it is observed that size and shape of the implant can affect the life of the prosthetic implant. The behaviour of the implant depends on various parameters. One can conclude from the literature reviewed that even the profile of the screw on implant can affect the service life and behaviour of the implant.

III. OBJECTIVE

To study the effect of different profiles of screw, pitch and diameters on the performance of the dental implant using finite element analysis.

VI. MATERIAL PROPERTIES

Material property	Titanium alloy	Cancellous bone
Modulus of Elasticity (MPa)	110000	0.1E3 to 2E3
Poison's Ratio	0.35	0.3
Yield stress (MPa)	936	100

V. DESIGN OF DENTAL IMPLANT

5.1. EXPERIMENTATION AND FINITE ELEMENT ANALYSIS OF IMPLANT:

It is observed from the literature that results obtained from numerical methods are to be validated. The dental implants are generally made up of titanium. In this dissertation work the efferent of the screw profile and the diameter are studied using advanced numerical method, finite element analysis.

5.1.1 Experimentation for load capacity of the dental implant:

The load capacity of the implant was tested on compressive testing machine. The implant was made up of titanium was used for the test. The implant placed in the compression testing machine. Following is the methodology for the experimentation and analysis if results, Dimensions of the implants are measured before the experimentation.

a. Using finite element analysis the load carrying capacity of only titanium was calculated.

b. The deformation at maximum load capacity was also noted down from the finite element analysis.

c. The load observed from the finite element analysis was used to load the implant.

d. The value of force is increased gradually.

e. At a point of load capacity the dimensions of implant are checked.

It was observed that values from finite element analysis are very near to that of experimentation.

5.1.2 Finite Element Analysis of Implant:

In ANSYS the Cad Model of Dental implant is imported. After that for analysis the Finite element model is generated. At the point of application of load the fine meshing is done. After meshing we get total 66031No. of Nodes and 65355 No. of Elements.

After proper meshing and boundary conditions the loading is to be assigned. Figure No. 2 Shows the Load and Boundary Conditions applied.

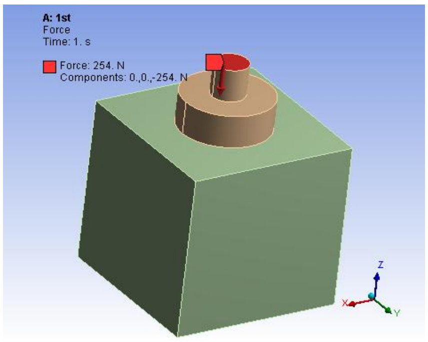


Fig.2 Load and Boundary Conditions

Table No .01 shows comparison between Load vs. Deflection from finite element analysis and Experimental Analysis.

Table 01. Comparison of Load vs. Deflection from finite element analysis and Experimental Analysis.

Load (kN)	Finite Element Analysis Deflection (mm)	Experimental Deflection (mm)
0.5	0.0062258	0.0061
1	0.012452	0.0121
1.5	0.018677	0.01815
2	0.024903	0.0242
2.5	0.031129	0.03132
3	0.031129	0.03312
3.5	0.04358	0.04351
4	0.049806	0.05002
4.5	0.056032	0.05671
5	0.062258	0.06325
5.5	0.068483	0.06894
6	0.074709	0.07505

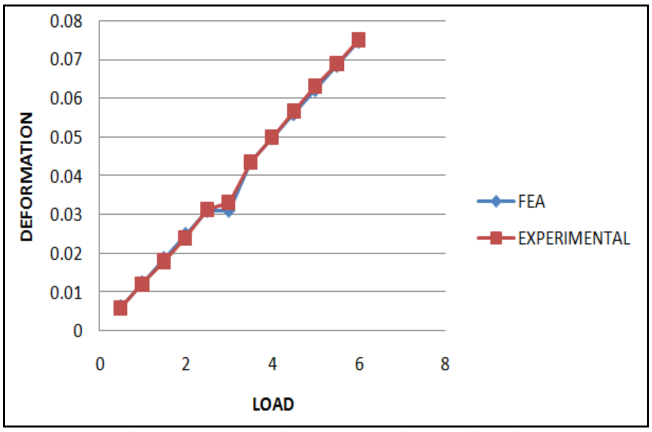
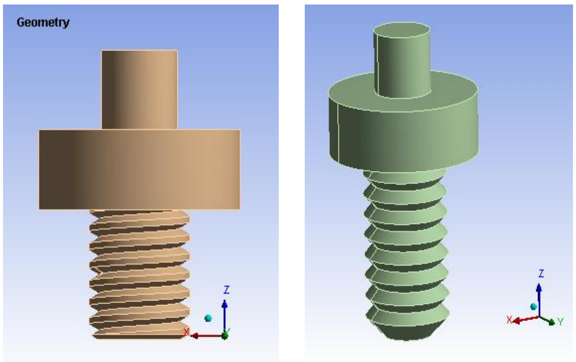
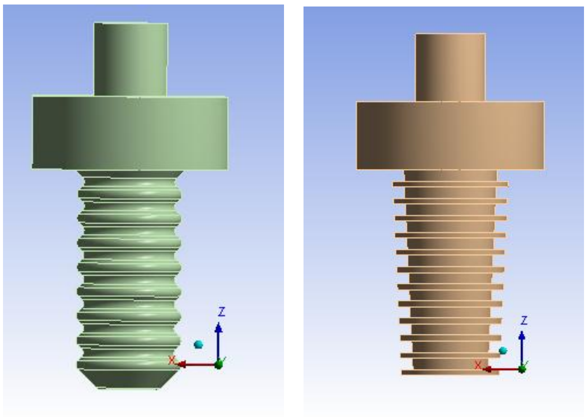


Fig.3 Graph for Comparison of Load vs. Deflection from finite element analysis and Experimental Analysis.

After the validation of results obtained from finite element analysis. Different screw profiles of dental implants are evaluated for strength and stresses.



(A) (B)



(C) (D)

Fig.4 Different screw profiles considered for analysis. A) Screw shaped ordinary, B) Conical profile, C) Curved Profile, D) Tapered diameter Implant.

Table 2. Comparative stresses and deformation in different screw profiles for dental implants.

Dental implant screw profile	Equivalent stress (MPa)	Deformation (mm)
Screw shaped ordinary	38.569	0.0047811
Conical	39.233	0.0038013
Curved	38.843	0.0037759
Tapered diameter	37.39	0.0037942

VI. Conclusions

From the above results it can observe that the implant with tapered diameter proved with the optimum results. As discussed earlier the design of implants cannot be done using empirical relations. Use of modern techniques proves helpful for design such complex problems. The implant with tapered diameter showed optimum results from the designs selected.

The Dental implant with tapered diameter has taper which helps the surgeon in fitting the implant. Also the stress concentration caused near the interfacing of the implant and the bone is also observed to be less as compared to others.

ACKNOWLEDGEMENT

The satisfaction and exhilaration that accompany the successful completion of any task would be incomplete without the mention of the people whose constant guidance and encouragement aided in its completion. The authors would like to express the voice of gratitude and respect to all who had directly or indirectly supported for carrying out this study and special thanks to Dr. B. M. Shinde, Asst. Prof., Mechanical Dept., DOER, Narhe, Staff of mechanical department, Prof. B. D. Aldar, HOD, Mechanical department and Dr. A. N. Gaikwad. Principal, Dnyanganga College of Engineering and Research,Narhe, Pune.

REFRENCES

[1]Lucie Himmlova, TatjanaDostalova, AloisKacovsky and SvatavaKonvickova, "Influence of Implant Length and Diameter on Stress Distribution: A Finite Element Analysis", The Journal of Prosthetic Dentistry, Volume No.91, Issue-1.pp.-20-25.

[2]Chun-Li Lin, Yu-Chan Kuo, Ting-Sheng Lin, "Effects Of Dental Implant Length And BoneQuality On Biomechanical Responses In BoneAround Implants: A 3-D Non-Linear FiniteElement Analysis" Vol. 17 No. 1 February 2005.

[3] M. Sevimay, F. Turhan, M.A. Kiliexarslan and G. Eskitascioglu, "Three-Dimensional Finite Element Analysis of the Effect of Different Bone Quality on Stress Distribution in an Implant-Supported Crown" The Journal of Prosthetic Dentistry, Volume No.93, Issue-3.pp.-227-234

[4] T. Li, L. Kong, Y. Wang, K. Hu, L. Song, B. Liu, D. Li, J. Shao, Y. Ding, "Selection Of Optimal Dental Implant Diameter And Length In Type Iv Bone: A Three Dimensional Finite Element Analysis", International. Journal Oral Maxillofacial Surgeons 2009; 38: 1077–1083.

[5]Jose Henrique Rubo, Edson Antonio Capello Souza, "Finite-Element Analysis of Stress on Dental Implant Prosthesis" Clinical Implant Dentistry and Related Research, Volume1,2009

[6]EkachaiChaichanasiri, PruetthaNanakorn ,WichitTharanon Jos Vander Sloten,"Finite Element Analysis of Bone around a Dental Implant Supporting a Crown with a Premature Contact" J Med Assoc Thai Volume 92, No. 10, 2009, pp-1336-1344

[7]B. AlperGultekin, Pinar Gultekin and SerdarYalcin,"Application of Finite Element Analysis in Implant Dentistry", Finite Element Analysis-New Trends and Developments, pp 22-54

[8]V. R. Citarella, E. Armentani, F. Caputo3 and M. Lepore,"Stress Analysis of an Endosseus Dental Implant by BEM and FEM", The Open Mechanical Engineering Journal,2012,6, pp-115-124

