International Journal of Emerging Science and Engineering (IJESE)
ISSN: 2319–6378, Volume-2 Issue-4, February 2014

A Review: Evaluation of Design Parameters of Dental Implant Abutment

Shraddha S.Mandhane, Amol P. More

Abstract— ***Dental implant is used to hold the artificial tooth into its proper position in human jaw. It serves the purpose of natural root which is there in natural tooth. Most of the dental implants are parallel or slightly tapered in shape but not as tapered as the natural roots. But due to some space and accessible constraint, it is not possible to provide taper same as that of the natural root. Failures of implant–abutment connections are relatively frequent clinical problems. So there is a need of analysis of dental implant abutment. For that research has been done on the existing design with its limitation. This paper focuses on Analysis and techniques used for evaluation of Dental Implant.***

Keywords— ***Dental Implant, Design Parameter, Evaluation, Analysis.***

I. INTRODUCTION

Loosening and fracturing of abutment and retaining screws appear to be significant problems in the application of dental implants. Fracture may occur if a screw is fatigued or overloaded, a condition likely exacerbated by loosening. Loosening, which takes place if a screw slips and detaches from its abutments. Now a day cad/cam system is implemented in most of the industries due to its accuracy and less time required for manufacturing the parts. So it is growing need to use cad/cam technology in dentistry which involves the use of complicated shape implants, abutments etc. Hence we decided to concentrate on one of the part i.e. dental implant. So there is need of optimization of existing design of Dental Implant Abutment. The objective of our research will be to obtain design of implant which is to be used for dental purpose by which screw loosening of dental implant will be minimum by considering washer and all other constraints.

II. GENERAL DESIGN OF DENTAL IMPLANT ABUTMENT

Tapered dental implants with tapering degree of 0.02, 0.06, 0.1, 0.12, and 0.16; and cylindrical implant with similar length (13.6 mm) and diameter (4 mm) were used; the abutment height was 6 mm. Implant pitch for all implants used is shown in Figure 1. No thread was modelled at the cortical bone level.The specific features of the implant abutment connections of each implant system schematically shown in figure.2,3,4. Commercially packaged implants and abutments were used.

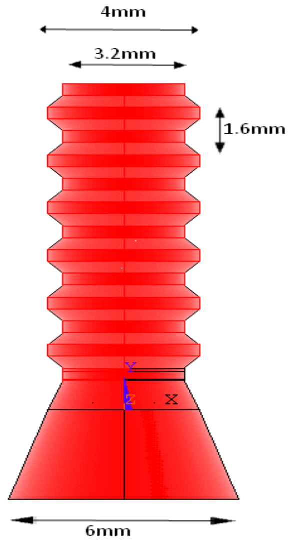

Figure 1 . Implant dimension

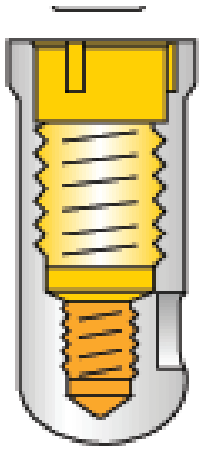

Figure 2.Implant

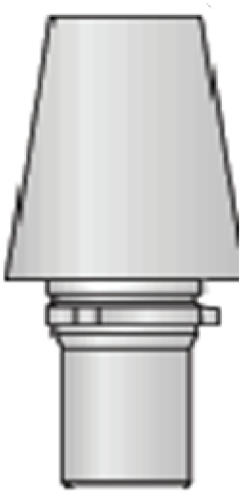

Figure 3.Abutment

Manuscript received February 15, 2014.
Shraddha S. Mandhane, M.Tech CAD/CAM, G.H.Raisoni College of Engineering, Nagpur, India.
Amol P. More, Asst. Prof (Mechanical Department), G. H. Raisoni College of Engineering, Nagpur, India.

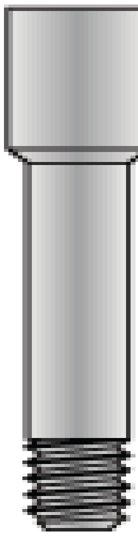

Figure 4.Abuutment Screw

III. PROBLEM STATEMENT

"To study the existing dental implant design and to find out the drawback in those design. Select optimize technique for Dental Implant analysis."

IV. OPTIMIZATION PROCESS

In addition to the parameter evaluated in the present study, additional factors must be taken into account when interpreting FEA .Material properties chosen for FEA have been shown to greatly influence the results obtained. Considering bone an anisotropic material significantly affects peri-implant stress and strain resulting from implant loading.Most frequently, results obtained from FEA simulations are reported as von Mises equivalent stress. Within the limitation of this study, it can be concluded that rather simplistic FEA model based on combinations of free-form object allows for proper estimitaion of stress mangnitude in bone that result from changes in geometric parameters, these difference appear to be less important for globle consideration. In that context,overall dimensions ,such as span length of a fixed restortion,seems to have prevailling effect on outcomes over more complex factor,such as thread geometry or the shape of supporting bone.[1]

The development of a new prosthesis designed to treat problem through new fixation geometry was based on the static loading and finite element analysis. To understand the behaviour of stress and its distribution along the implant, different types of implant, threaded cylindrical implant and spherical-lobe implant (solid-lobar implant) and hollow-lobar implant are analysed. The ANSYS Finite Element Package was used. The resulting 3-D model volumes were discredited using 8-node explicit brick elements. The bone-implant interface was considered fully bonded. Axis symmetric FE models are constructed for all implant-abutment-bone systems. The implants and abutments are modelled as Ti6Al4V with linear-elastic, isotropic and homogenous properties. Based on analysis and discarding the models with higher values of implant stresses, entering the new design parameter shows a good compromise between the stress shielding and mechanical performance of the dental implant prosthesis according to the finite element analysis results.[2]

Finite elemental analysis is an efficient technique for investigating biomechanical interactions of different implant designs. To investigate the effect of cylindrical and tapered implants with different degree of tapering and similar lengths on the stress and strain distribution in the bone and implant. One cylindrical and five types of tapered implants with degrees of tapering from 0.02 to
0.16 were modelled to be used . The implant material was grade IV titanium and abutment was grade ELI titanium. The bone model used comprised of compact and spongious bone assumed to be homogeneous, isotropic and linearly elastic The generation of high stress distribution or concentration in the bone should be avoided to achieve stable osteointegration for implant restoration; therefore, the influence of implant on stress and strain distribution in the bone must be investigated. In this study, the effect of implant on stress distribution in the bone under vertical pressure was investigated by performing finite element analysis using Ansys computer software with contact friction at the interface between the abutment and implants. Maximum von Mises stress in the bone occurred at the region of cortical bone adjacent to implant neck and lower bound of cortical bone. With increased degree of implant tapering, the von Mises stress and strain increased in the bone. However, at the neck of implant, the most sensitive area, with increase in degree of tapering, both stress concentration and strain decreased. The lowest stress and strain were generated in the most tapered implant.[3]

The complicated relationship between mandibular bone components and dental implants have attracted the attention of structural mechanics researchers as well as dental practitioners .Using the finite element method, the present study evaluated various bone and implant parameters for their influence on the distribution of von Mises stresses within of cancellous bone, whitch varies from 1 to 4Gpa,and that of cortical bone ,which is between 7 and 20 Gpa.Implant length (7,9,11,13, and 15mm), implant diameter (3.5,4.0,4.5 and 5.5mm),and cortical bone thickness (0.3 to 2.1mm) were also considered as parameters.b Assumptions made in the analysis were;modelling of the complex material and geometric properties of the bone and implant using two-dimensional triangular and quadrilateral plane strain elements. An increase in Young's modulus and a decrease in the cortical bone thickness resulted in elevated stresses within both cancellous and cortical bone. Increases in the implant length led to greater surface contact between the bone and implant, thereby reducing the magnitude of stress.[4]

In this paper the author compared the loosening torque of experimental conical-head abutment screws to that of conventional flat-head screws of implants with external-hex (EH) and internal tri-channel(IT) connections before and after mechanical loading. Abutments were tightened at 32 Ncm of torque; after 10 minutes, loosening torque was measured. The same abutments were then retightened with 32 Ncm of torque; after 10 minutes, they were mechanically loaded for 300,000 cycles and loosening torque was again measured. Data were collected and an exploratory analysis was performed. The shape of the abutment screw head significantly influenced loosening torque; conical-head screws showed higher loosening torque values than conventional flat-head screws before and after loading. The implant/abutment connection design exerted no significant influence on loosening torque. [5]

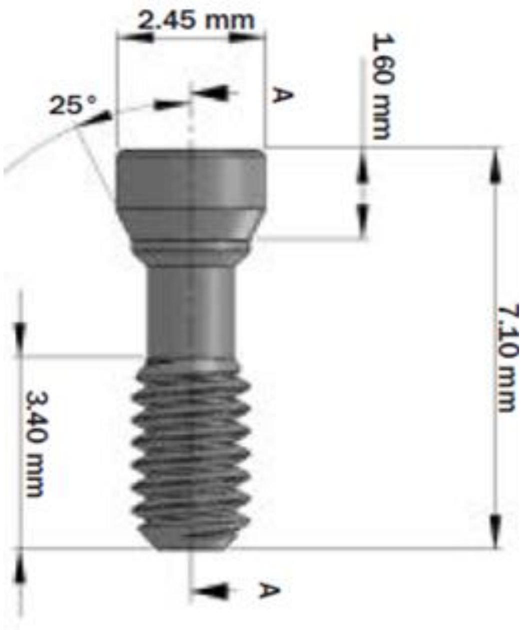

Figure.5 Conical screw for EH Abutment

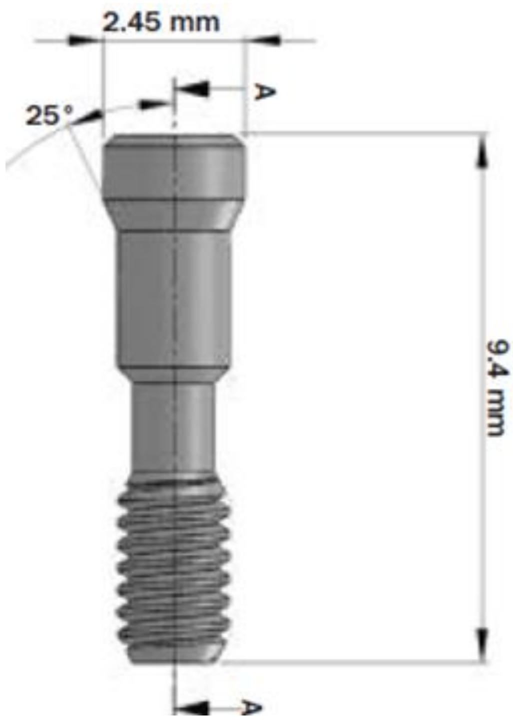

Figure.6 Conical Screw for IT Abutment

The implant length and diameter also have a significant influence on the stress distribution at the bone-implant interface. Finite element analysis of stress distribution around implants can be used to determine the optimum length and diameter of the implants. Von Mises stresses at the bone-implant interface were calculated using FEA for different diameter and length of implants. Implant neck was identified as the most critical area having maximum stress distribution. The minimum stress was found for implants with a diameter ranging from of 3.6 mm to 4.2 mm. Increase in the implant length also decreased Von Mises equivalent stress values. Influence of implant length was less as compared to implant diameter. Short implants present superior failure rates. A lot of research has been done on different parameters of dental implants which affect the stress distribution at the bone-implant interface. Very minimal research to find the optimum inter implant distance has been carried out. Research on the above mentioned points can prove to be very vital in dental implant design.[6]

To obtain geometrical details in micrometer range, special manufacturing processes are required. Micro casting is one of the key technologies enable the manufacture of small structures in the micrometer range or of larger parts carrying microstructures by using a metal melt which is cast into a micro structured mold. Micro casting, is generally identified with the investment casting process, which is known as the lost-wax, lost-mold technique. The process has the advantages of reproducing complex shapes at relatively low cost, scalability from single items to large numbers of identical items, and low wastage of raw materials.

First a plastic or wax pattern is made and is coated with a nonreactive face coat intended to minimize the reaction between the ceramic shell and the molten metal during casting. Then the pattern is dipped in a ceramic slip repeatedly. It is essential to maintain a uniform wall thickness in the shell both for mechanical strength and for heat transfer during solidification. After drying the ceramic mold is heated and sintered and the pattern will be lost during this process due to melting and burning. Finally the preheated ceramic mold is filled with metal melt by vacuum-pressure or centrifugal casting. After solidification, the ceramic mold is mechanically removed without destroying or influencing the cast surface. Finally, the single parts are separated from the runner system. At last to cope the challenges about strength of implants under different kinds of loadings while using, modeling and analyzing with finite elements softwares such as other analysis that have been done by this type of softwares can be done.[7]

In this paper , a new three-dimensional(3D) method of evaluating the fit of implant superstructures are using computer-aided design/computer-assisted manufacture (CAD/CAM) technology and conventional casting and to determine which biomaterial would produce optimal fit for the long-term clinical longevity of dental implant restorations. When used in combination with the CAD/CAM technique , titanium produces the most accurate implant superstructure. Spiral scan microtomography can be used to measure the accuracy of fit of dental implant superstructures and restorations as it provides a 3D measurement with less chance of errors compared with conventional methods of measurement.[8]

An implant macro design includes thread, body shape, and thread design [e.g., thread geometry, face angle, thread pitch, thread depth (height), thickness (width), or thread helix angle]. Thread shape is determined by the thread thickness and thread face angle. Thread pitch refers to the distance from the center of the thread to the center of the next thread, measured parallel to the axis of a screw.[6]

Implant threads should be designed to maximize the delivery of optimal favourable stresses while minimizing the amount of extreme adverse stresses to the bone–implant interface. In addition, implant threads should allow for better stability and more implant surface contact area. Although the thread pitch and depth could affect the stress distribution, traditionally, the manufacturers have provided an implant system a constant pitch and depth. So, for the commercial implant system, a better design of thread configuration is emphasized.[9]

V. CONCLUSION

From the review of Research paper it is concluded that proper implant design is crucial to ensure long-term fatigue performance for dental implants. The combination of sharp notches(thread) and narrow metal cross section might be deleterious for fatigue resistance implant. But more Evaluation techniques need to find for better design of Dental Implant.

REFERENCES

1. Wener Winter, Stefan Holst "Effect of geometric parameters on finite element analysis of bone loading caused by nonpassivly fitting implant-supported dental restoration" Quintessence International 2011;42:471-478.
2. Hussam El-Din El-Sheikh " Design Developeent of an Endosseous Dental Implant " J Prosthet Dent, 92:523-530, 2004.
3. Bijan Heidari, Hossein Bisadi , Behnam Heidari And Mahdi Kadkhodazadeh " Influence of Different Tapered Implants on Stress and Strain Distribution in Bone and Implant: A Finite Element Analysis" J Periodontol Implant Dent 2009; 1(1):11-19
4. Hong Guan,Rudi Van Staden,Newell Johson "Influence of Bone and Dental Implant Parameters on Stress Distribution in the Mandible: A Finite Element Study" Int.J Oral Maxillofac Implants 2009;24;866-876
5. Abilio Ricciardi Coppede,Adriana Claudia Lapria Faria,Jail Awad Shibli "Mechanical Coparison of Experimental Conical-Head Abutment Screws with Conventional Flat-Head Abutment Screws for External-Hex and Internal Tri-Channel Implant Connections:An In Vitro Evaluation of Loosening Torque" Int.J Oral Maxillofac Implant 2013;28:e321-e329.
6. Sarthak Seth, Parveen Kalra "Effect of Dental Implant Parameters on Stress Distribution at Bone-Implant Interface" -International Journal of Science and Research (IJSR), India Online ISSN: 2319- 7064 volume2 Issue 6,2013.121-124.
7. Payam Faghihi1, M. M. Mohammadi2, Dr. M. Besharati Ghivi "Fabrication of Mini and Micro Dental Implants using Micro Investment Casting and Its Challenges" International Journal of Science and Engineering Investigations ISSN: 2251-8843 ;2013;97-101
8. Rahul Prasad , Abdulaziz Abdullah Al-Kheraif "Three –Dimensional Accuracy of CAD/CAM Titanium and Ceramic Superstructures for Implant Abutment Using Spiral Scan Microtomography" Int J Prosthodont 2013;26;451-457.
9. SR Desai, MS Desai1, G Katti2, I Karthikeyan "Evaluation of design parameters of eight dental implant designs: A two-dimensional finite element analysis" Nigerian Journal of Clinical Practice ; 2012 ; 15(2);176-181.
10. Steinebrunner L, Wolfart S, Ludwig K, Kern M " Implant–abutment interface design affects fatigue and fracture strength of implants" Clinical Oral Implant, 2008; 1276–1284 .

AUTHORS PROFILE

Ms. Shraddha S.Mandhane, is Research Student of M.Tech in CAD/CAM at G.H.Raisoni College of Engineering, Nagpur, India.

Amol P. More, is working as Assistant Professor in Mechanical Engineering Department at G.H.Raisoni College of Engineering, Nagpur, India. His research of interest is CAD/CAM.