A review: Mechanical Design of Dental Implants to Reduce Stresses around Implant-Bone Interface

Rajiv. S. Jadhav
NBN Sinhgad School of Engineering, Ambegaon (Bk) Pune, India
jadhav.rsa@gmail.com

Prof. P. N. Dhatrak
MAEER's Maharashtra Institute of Technology, Pune-411038.
pankajd123@gmail.com

Dr. S. Y. Gajjal
NBN Sinhgad School of Engineering, Ambegaon (Bk) Pune, India.
shekhar.gajjal@sinhgad.edu

Abstract: Now-a-days finite element analyses are the powerful tools which is been used in the implant dentistry. In this study, it was been reviewed that Finite element analysis were performed on various thread shapes of dental implants of different manufacturers to study effects on stress distribution generated in the surrounding jaw bone and to determine an optimal thread shape for even stress distribution in jaw bone. The aim of this study was to derive alternative implant thread shapes which could minimize the stress concentration at the shoulder level of the implant. A literature survey was done with the index terms mentioned below. Several relevant articles and research papers were studied for the collection of the data which is used for the further work. Different parameters like types of implant thread designs, implant materials and properties , loading conditions, methodologies is studied. The implant thread designs are been altered to get the expected results. The study of the implant thread parameters is been carried out further for the research work

Keywords: Dental implants, Finite element analysis, Kinds of threads, Stress, Thread pitch, Thread design, Thread parameters.

I. INTRODUCTION

What is dental implant? Dental implant is a biocompatible part, an artificial tooth which is a prosthetic replacement for a missing tooth. Basically the natural teeth consist of two parts viz. crown and a root. The crown is the visible part that is covered with white enamel and supporting the crown is the tooth root which extends into the jaw bone. The root is effectively replaced by an implant with the surgical method. Implant is inserted directly into the mandibular or maxillary bone. Today implants are widely used throughout the world for replacement the decayed or with prosthetic supports teeth. There are several factors involved in the failure of implant like poor oral and dental health, the type of loading on dental implant, quality and quantity of the bone around it, surgical procedures, post-operative care and implant geometry. Also several factors have effect on the stability of the implant including type and design of the thread. How does the implant thread play an important role in the stress distribution in the jaw bone is been studied and reviewed.

In this review, we collected the data regarding the effect of the thread design on stress distribution, optimum thread design, loading and boundary conditions, type of analysis, assumptions made, results and discussion**.**

Figure: Dental Implant

II. DESIGN PARAMETERS OF THREADED DENTAL IMPLANT

Design of dental implant includes the study of various parameters like overall geometry of threaded implant, types of threads, thread pitch, etc. so that stress is uniformly distributed around implant surface. There are different types of the threads in the implants which are been used accordingly.

Liang Kong [1] in his research focused on effects of the implant thread pitch on the maximum Von Mises stresses. The thread pitch were varied and the optimum thread pitch was been selected that gave the minimum stresses. Thread pitch was set as an input variable and maximum EQV stress was considered as the output variable. He studied the thread pitch ranged from 0.5 mm to 1.6 mm. When thread pitch exceeded 0.8 mm, minimum stresses were obtained. [10, 11, 12]

Oğuz Eraslan [2] in his study has aimed to evaluate the effects of different implant thread designs on stress distribution characteristics at supporting structures. In his work he simulated four types of 3D mathematical models simulating four different thread-form configurations for a solid screw implant with supporting bone structure. V thread, buttress, reverse buttress, and square thread designs were simulated.

Zeinab Arsalanloo [3] has tried to find that how thread types can affect the amount, type of load and biomechanical responses induced in mandible/maxilla and implant–abutment complex by a finite element method. She has also explained that implants with extended threads are highly recommended in implant dentistry today to enhance initial stability, enlarge surface contact area, favor dissipation of interfacial stress, and reduce micro movements of the implant during post insertion healing period until the stable osseo-integration is established. [13]

According to H.J. Chun [4] he performed finite element analyses for various thread shapes of dental implant to study effects on stress distribution which was generated in the mandibular bone and to find an optimal thread shape for even stress distribution. He investigated that the square thread shape filleted with a small radius was more effective on stress distribution than other thread shapes used in the analyses. He performed some additional analyses on the implant with the thread shape obtained from previous analyses for varying other thread parameters, like the width of thread end and height of thread for various load directions, to determine the optimal dimensions of the implant. He found that stress distribution was more effective in the case when the width of thread end and the height of thread were 0.5p and 0.46 p, respectively, where p is the screw pitch. Further, using the optimal implant thread dimensions found previously, the stress analyses were performed with various screw pitches

Iman Zarei [5] studied the purpose of investigating the geometry and type of thread of implants and their function. Type and design of the thread is one of the factors that have effect on the constancy and stability of the implants. In this study the effect of design of different types of implants with different thread has been examined and distribution of their stress on surrounding bones of implant has been compared.

III. MATERIALS AND METHODS

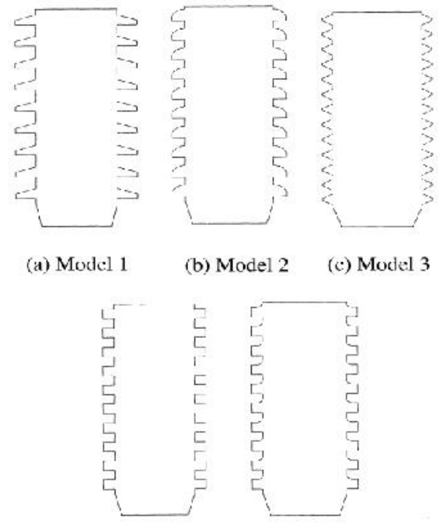
Model design-

Liang Kong [1] in his study modeled a mandible segment with a screwed implant and a superstructure on a personal computer, using a 3D modeling software (Pro/E Wildfire, Parametric Technology Corporation, USA). He modeled a cross-section of a mandible in the first premolar region as a basis of solid model. The cross-sectional image was then extruded to create a three-dimensional mandible segment, which contained a thick layer of cortical bone surrounding the dense cancellous bone

Oğuz Eraslan [2] in his research used Solidworks for FE modeling. The study was conducted using a 3D Solidworks 2007 9.0.3 structural analysis program (Solidworks Corporation, Concord, MA, USA). An ITI solid cylindrical screw implant, 3.8-mm diameter, 10-mm bone sink depth (Straumann AG, Waldenburg, Switzerland) was modeled. Materials used in study were assumed to be homogenous and isotropic. Elastic properties of materials (Young's modulus (E) and Poisson's ratio (μ) were determined from the literature [13]

There were 26 different implant designs of 7 implants company categories, used in this study by Zeinab Arsalanloo [3], with the diameter ranging from 2 to 3.5 mm. This series of diameters is called Narrow Platform series and some other with diameters smaller than 3 are called mini-implants. Among the different implants, the length of the implants varies from 8 to 16.0 mm for upper/lower lateral incisor situation. Lower jaw section geometries in lateral incisor tooth position are of height 88.461 mm, width of 48.514 mm, thickness of 10 mm. Jaw bone consist of cortical and cancellous bone. The cortical bone is outer layer of jaw. In this research, the author has an assumption that materials are linear elastic isotropic for Titanium alloys and isotropic for Cobalt-Chrome alloys. The implant, abutment, and abutment screw were all designed to be Titanium. As bone is considered as a porous structure, so they used nonlinear isotropic properties

H.J. Chun [4] in his study selected, commercially available implants with different thread shapes for analyses, the plateau type (model 1), plateau with small radius of curvature (model 2), triangular thread screw type with 0.7 mm in screw pitch in accordance with ISO regulations (model 3), square thread screw with 0.9 mm in screw pitch in accordance with ISO regulations (model 4) and square thread screw filleted with small radius partially (model 5)



(a) Model 1 (b) Model 2 (c) Model 3
(d) Model 4 (e) Model 5

Figure: Different thread shapes [4]

The thread of implant was modeled as symmetric. Jaw bone of 20 mm in height and 16 mm in diameter

Iman Zarei [5] has studied a three-dimensional model of the bone using CBCT (Cone Beam Computerized Tomography) of a 55 year old patient by Mimics 10.01 software. The created three-dimensional model of bone was exported to Solid Works software for cleanup. The jaw bone was considered with two layers of cortical (thickness 0.75 mm) and trabecular. He used Implants from DENTIS (Dental Implant Solution) Company for design. In this research the implants with the length of 10 mm and diameter of 4.1 mm (manufacturer's catalog) are used.

IV. FINITE ELEMENT ANALYSIS

Liang Kong [1] performed a three-dimensional finite element analysis on variation of implant thread pitch and to determine the optimal implant biomechanical parameter Maximum Von Mises stresses were evaluated in jaw bones and implant–abutment complex by a finite element method. All models were meshed and analyzed by ANSYS Workbench10.0 (SAS IP, Inc., USA). [1] Liang Kong in his study meshed the models with 10-node-tetrahedron and 20-nodehexahedron elements. The mesh was made finer around the implant. Models were composed of 170,000 elements and 250,000 nodes in average.

Oğuz Eraslan [2] in his research used Cosmos works structural analysis programs for FE analysis. He meshed the geometric models with tetrahedral quadratic elements. Each mathematical model included approximately 186,000 nodes and 133,000 solid elements. The bottom exterior nodes of the alveolar bone in the FEM models were fixed in all directions as the boundary condition. A 100-N static axial occlusal load was applied to occlusal surface of abutment to calculate the stress distributions [13].

Zeinab Arsalanloo [3] has introduced three dimensional Finite element technique for getting the optimum conditions. In this study, a non-linear and complex static analysis was been performed The element in meshing all three-dimensional models is 8 noded Brick element (SOLID45), which has three degrees of freedom The interface between implant and bone was modeled as an immovable and rigidly junction, which simulated the condition of the optimal implant osseointegration The bone and implants simulated models were meshed tetrahedron elements. A finer mesh was generated around the implant.

H.J. Chun performed a two dimensional finite element analysis on various thread shapes to find out the optimal thread shape which has a more even stress distribution in the jaw bone. It was assumed for boundary conditions that interface between implant and jaw bone was a perfect bond and that the edges of bone were fixed horizontally and vertically for all analyses. In his study, both bone and implant were assumed to be homogeneous, isotropic and linearly elastic. The loads applied were 100 N vertically and 100 N with 15 degree angle. The downsized elements were used at the locations where the higher stress level was expected. As the stress changes drastically at the border between implant and bone it was meshed three times as dense as the edges, The number of nodes and elements for the models in this study averaged a total of 39 000 and 13 000, respectively [4][13]

Iman Zarei [5] in his study used Solid 187 elements for meshing of various parts. Solid 187 element is a quadrilateral element with 10 nodes used for meshing the complex geometry in three dimensional analysis. The total number of elements and nodes were 725,046 and 1,133,167 respectively and the average of dimension size of any element was considered 0.15 mm. In this study, boundary condition for the upper jaw was considered completely fixed. As the size of elements is smaller and the number of them is more and general shape of meshing is more regular, stress distribution will be more accurate and reliable, and consequently the performed analysis will be more valid and accurate

V. RESULTS

Liang Kong [1] analyzed that when pitch was more than 0.7 mm, the most stable and minimal stress level could be achieved in cortical bone. He also studied that the most stable stress and minimal stress level in cancellous bone and implant–abutment complex were found when pitch was more than 0.8-mm and over 0.75-mm, respectively

Oğuz Eraslan [2] found that the analysis of the Von Mises stress values revealed that maximum stress concentrations were located at loading areas of implant abutments for all models. Also, high stress values were located at cervical cortical bone regions adjacent to implants at all models. It was seen that the stress concentration at cortical bone structure was higher than that of spongious bone at 3D model view. He also learned that the stress concentration of bone structure adjacent to first thread which was located at cortical bone structure was higher than that of other threads which were located at spongious bone.

Zeinab Arsalanloo [3] has provided a detailed discussion and results to find out pure effect upon the stresses of variations of the thread shapes. She gathered the data about the thread parameters like shape, depth, pitch, and angle of threads. In this paper it was found that bone defects tended to be located at the thread tops. Now-a-days, newest threaded implants try to use a combination of various shapes for thread.

H.J. Chun [4] found that the maximum effective stress under both loading conditions occurred at the regions in jaw bone adjacent to the first thread of implant and maximum effective stress induced by the oblique load was twice as high as the maximum effective stress caused by an equal amount of vertical load. He also learned that maximum effective stress in the model with square thread filleted with small radius, was the lowest when compared with that in all models and stress distribution in the same model was more even than that in other models used in the analyses. He also found that the maximum effective stresses generated in the surrounding jaw bone with a vertical load and a 15 degree oblique load of 100 N were 13.256 and 29.4 MPa, respectively

Iman Zarei [5] has found that by changing thread, the differences were observed in amount and manner of the stress distribution. The maximum stress has happened at the first contact point of implant with the bone**.**

VI. DISCUSSION

Liang Kong [1] in his study discussed that the influence of the threads can be easily understood. Greater the number of threads also greater the depth of the threads, the more functional surface area is available. In the cortical bone, the thread number changed less than that in the cancellous bone. This may be the important reason for that thread pitch favoring stress distribution in cancellous bone than that in cortical bone. Furthermore, the implants with over-sized thread pitch (>1.6 mm) are believed to thread faster into the bone site causing higher stress in jaw bone and implant, which also can go against the operation of dental surgeon. It is recommended to avoid design the implant with over-sized thread pitch.

Oğuz Eraslan reported in the literature that stress (compressive) was more evenly distributed when the implant thread shape was square than V-thread shape

Zeinab Arsalanloo [3] has discussed about the thread parameters like pitch, thread angle, depth of thread. The author has explained that if the pitch is increased, the numbers of thread on implant is decreased thus reducing the amount of stress on threads. But stress reduction on threads may cause the lack of initial stability of implants. The included angle of thread describes the threads type. Threads are a combination of V shape, Acme, Buttress and German Buttress. The author has investigated that increase or decrease in depth of thread can change the amount of the stress

H.J. Chun [4] in his research work discussed that the contact area between implant and the jaw bone increased with the decrease of screw pitch. Stress decreases with the decrease in pitch

Iman Zarei [5] in his study has performed two or three-dimensional finite element analysis. Two or three-dimensional analysis is been done depending on the complexity of the model, the rate of required accuracy, application of the results and complexity of structures in the analysis. In general, three-dimensional models show human anatomy and biomechanical interactions better. Meijer in his study has recommended that the two-dimensional model should not be used for analysis of stress in implant. Van Staden in his study mentioned that the choice of two or three-dimensional analysis depends on the type of software but according to the complexity of structure of the jaw and dental implants, to obtain a reliable and accurate study, it is better to use a three-dimensional finite element analysis [7, 6]. Abu Hussein and Kong [8, 9] in their study showed that the optimal depth and width of thread of implants are between 0.34 to 0.5 and 0.18 to 0.3 mm and estimated that the optimal length of thread pitch is 0.8 mm and they believe that the pitch length less than 0.8 mm attracts more stress to itself.

VII. CONCLUSION

Based on the literature, we have drawn the conclusions related to the implant thread shapes and thread parameters like pitch, depth of thread and their relation with the stress distribution in surrounding jaw bone

1. We can conclude that thread configuration is one of the major contributors to initial implant stability. Further initial implant stability will promote successful osseointegration and successful osseointegration will lead to distribution of stress over wide area and at low level. By Increasing the pitch of the thread, number of threads are been reduced but that hampers badly on the initial stability of implant.

2. Thread pitch plays a great role in protecting dental implant under axial load than under buccolingual load. Thread pitch exceeding 0.8-mm is optimal biomechanical property for the type B/2 bone in a cylinder implant, but over-sized pitch should be avoided.

3. Different implant thread forms produce different compressive stress intensities at bone structure. Cortical bone and bone structure adjacent to first thread bears more both von Mises and compressive stresses than spongious bone.

4. The highest stress concentration occurred at the region in jaw bone adjacent to the first thread of implant in finite element analyses. The maximum effective stress induced by an oblique load was twice as high as the maximum effective stress caused by an equal magnitude of vertical load. Stress was more evenly distributed in the case when the implant shape was square thread filleted with a small radius. Stress distribution was most effective in the case when the width of thread end and the height of thread were p/2 and 0.46p, respectively. Where p is pitch

5. Changing screw pitch is more effective way for reducing maximum effective stress, as changing pitch is more feasible way if considered from economical point of view and biological parameters

6. The implant with smaller threads and less than l mm pitch length causes increase of the stress in the bones adjacent to the implant.

7. The implant with deeper thread because of having more contact surface with the bone has more stability and less stress and also enhances the stress distribution

8. The upper threads of implant that are near to the neck of implant, in comparison with the threads in bottom, have more important role in distribution of stress, consequently, this part should be more considered.

REFERENCES

[1] Liang Kong a, Yunzhuan Zhao a, Kaijin Hua, Dehua Li b, Hongzhi Zhou a, Ziyan Wuc, Baolin Liu a. Selection of the implant thread pitch for optimal biomechanical properties: A three-dimensional finite element analysis. Advances in engineering software 40(2009) 474-478

[2] Oğuz Eraslan & Özgür İnan. The effect of thread design on stress distribution in a solid screw implant: a 3D finite element analysis. Clin Oral Invest (2010) 14:411–416
[3] Zeinab Arsalanloo, Reza Telchi, and Kambiz Ghaemi Osgouie. Selection of Optimum Thread Type in Implants to Achieve Optimal Biomechanical Properties by Using 3D Finite Element Method. International Journal of Bioscience, Biochemistry and Bioinformatics, Vol. 4, No. 3, May 2014. 185-190
[4] H.J. Chun, S.Y. Cheong, J.-H. Han, S.J. Heo, J. P. Chung, I. C. Rhyu, Y. C. Choi, H. K. Baik, Y. Ku & M.H. Kim. Evaluation of design parameters of osseointegrated dental implants using finite element analysis. Journal of Oral Rehabilitation 2002 29; 565–574
[5] Iman Zarei, Salar Khajehpour, Parham Zahedinejad. The Effect of Thread Design on Stress and Strain Distribution around Dental Implants, Using Three-Dimensional Finite Element Analysis. ISSN: 1300-1876 Vol.04, No.05 (2015)
[6] Meijer HJ, Starmans FJ, Bosman F, Steen WH. A comparision of three finite element models of an edentulous mandible provided two implants. J Oral Rehabil 1993; 20:147-157.
[7] Van Staden RC, Guan H, Loo C. Application of finite element method in dental implant research 2010.
[8] Abuhussein H, Pagni G, Rebaudi A, Wang HL. The effect of thread pattern upon implant osseointegration. Clin Oral Implants Res 2010; 21(2):129-36.
[9] Kong L, Liu BL, Hu KJ, Li DH, Song YL, Ma P. Optimized thread pitch design and stress analysis of the cylinder crewed dental implant. West China Journal of Stomatology 2006; 24, 509–12, 515.
[10] Ivanoff CJ, Grondahl K, Sennerby L, Bergstrom C, Lekholm U. Influence of variations in implant diameters: a 3- to 5-year retrospective clinical report. Int J Oral Maxillofac Implants 1999; 14:173–80
[11] Huang HL, Chang CH, Hsu JT, Fallgatter AM, Ko CC. Comparison of implant body designs and threaded designs of dental implants: a 3-dimensional finite element analysis. Int J Oral Maxillofac Implants 2007; 22:551–62.
[12] Misch CE. Contemporary implant dentistry. 2nd ed. St. Louis: Mosby; 1999
[13] S. Hansson and M. Werke, The implant thread as a retention element in cortical bone: the effect of thread size and thread profile: a finite element study. J. Biomech, vol. 36, pp. 1247-58, 2003.
[14] B. Alper Gultekin, Pinar Gultekin and Serdar Yalcin. Application of Finite Element Analysis in Implant Dentistry. Intech.24-54

About Authors:

Rajiv. S. Jadhav is a Post Graduate student in Mechanical Design Engineering in NBN Sinhgad School of Engineering, Pune under Savitribai Phule Pune University, Maharashtra, India)

Pankaj. N. Dhatrak is working as an Assistant Professor in Mechanical Engineering Department of MAEER'S M.I.T. Pune. He is working on interdisciplinary projects viz. Numerical and experimental investigations of stress distribution pattern between implant and bone interface, Effect of surface roughness on fatigue life of dental implants. His work experience spans more than 10 years in teaching and 4 years in Industry. He is pursuing his PhD from Savitribai Phule Pune University, Maharashtra, India. He is a life member of The Indian Society for Technical Education (ISTE- LM 99151)

Dr. Shekhar. Y. Gajjal is working as a PG Head and Professor in Mechanical Engineering Department of STES, NBN Sinhgad School of Engineering Pune. He has guided many PG projects in mechanical design engineering. He is BoS committee Member of Savitribai Phule Pune University, Faculty of Mechanical Engineering

.