A Review: Design Parameters of Dental Implants to Reduce Stresses Around Implant-Bone Interface

Mr. Durvesh Jitendra Thakur  
Department of Mechanical Engineering  
Saraswati College of Engg., Kharghar, Navi Mumbai, India  
E-mail: durvesh.thakur01@gmail.com

Prof. T.Z. Quazi  
Department of Automobile Engineering  
Saraswati College of Engg., Kharghar, Navi Mumbai, India  
E-mail: kazitaqui@rediffmail.com

Prof. Pankaj N. Dhatrak  
Department of Mechanical Engineering  
MAEER’S MIT, Pune, India.  
E-mail: pankajd123@gmail.com

Abstract: Now-a-day deep research is going on dental implants. Dental implants are used to hold the artificial tooth into its proper position in human jaw. Placement of implant is done by surgical procedure which is very complicated. During actual use of the implants stresses are developed around it. Finite elemental analysis is one of the powerful tool used for stress calculations around dental implants. In this paper, it was been reviewed that Finite element analysis were performed on dental implants to study the effects of various thread pattern on stress distribution around bone-jaw interface. 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 design parameters like thread pattern of implants, diameter, pitch, length, implant materials and properties, loading conditions, methodologies were studied. The study of the implant thread parameters is been carried out further for the research work. The aim of the study is to optimize the design parameters of dental implants to reduce the stress occurs around dental implants.

Keywords: dental implants, design parameters, Finite element analysis

I. INTRODUCTION.

Implants are the effective tool for replacing missing teeth. Implants are most successful but expensive treatments used in medicine. However, the number of failures is still relevant and limiting these failures remains one of the goals in today’s implant research. Generally implants having three detachable parts such as main implant body, abutment and fitting screw used in between them to fit the whole assembly. Dental implant body is fixed in human jaw and it is tapered and having thread patterns around its outer surface so it is become mechanical procedure to implant them into human jaw. Abutment is then going to fit over it by fitting screw, And finally the artificial teeth is placed on upper portion of the abutment.(Fig no.1)

Many factors have been found to influence this interfacial bonding between the implant and bone and thus the success of implants such as, surgical technique, implant design parameters, implant surface, material biocompatibility and loading conditions, quality and quantity of local bones etc. Here different approaches for analyzing stresses distributions around dental implants during its implantation and actual working condition were studied and reviewed. Most of the failures occur due to heavy stress causes just because of selection of wrong design features of implant and uneven load created during operation which turns to make unsuccessful implantation. Finite elemental analysis is one of the effective approach used for calculation of stress distributions around the dental implants and helpful for getting proper idea about uneven stress concentration around them. The aim of the study is to understand the effect of design parameters on stress distribution around dental implants and reduce the stress by making possible alteration in designs of implants. 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.

II. DESIGN PARAMETERS OF DENTAL IMPLANTS

Dental implant design parameters mostly includes the study of overall geometry of threaded implant which includes various threads patterns, thread pitch, diameter, length of implants etc. and these parameters are having individual effect on the stress distribution around dental implants.

Mohamed I. El-Anwar[1] in his research he has focused on study of stresses distributions and displacements of different threaded dental implant designs (diameter and length), which are considered as the most effective parameters on stresses distribution in surrounding bones (cortical and spongy). He has taken FEA approach to find the impact of variation in diameter and length of the implants on stress distribution around it. For his research he has taken Twenty-five different
implant designs placed in a cylindrical bone section along with variation in length and diameter.

Jenny Zwei-Chieng Chang[2] in his study he has examined 4 types of self-drilling mini-implants to find out the impact of thread depth, taper shape and taper length on mechanical properties of the implants such as insertion, torque, pullout strength, stiffness, and screw displacement before failure. His objective was to find optimal range of considered design parameters of mini-implants which gave improved mechanical properties [8]. He has conducted mechanical experiments on 4 no’s of Titanium-alloy (Ti6V4) mini-implants with fixed 2.0-mm external diameters, 9.82-mm thread lengths, and 0.75-mm thread pitches to validate FEA results for given considerations.

Liang Kong[3] in his research, he focused on the effect of dental implant pitch on maximum von-mises stresses generated around the implant assembly. His intension was to find optimal range of the pitch that gave minimum stresses and also provide protectiveness to the implants. For this, he has selected the pitch range from 0.5 to 1.6 mm and also found the individual impact on stress distribution using finite elemental analysis approach. Under his observation and study he came to know that when pitch exceeded beyond 0.8 mm would give minimum stresses.

T. Li [4] in his research aimed to identify the optimal range of the diameter and length of the dental implants in type IV bone under biomechanical consideration using finite elemental method. Here he studied the implants having diameter ranges from 3 to 5 mm and length ranges from 6 to 14 mm. He studied the impact of variation of diameter and length on stress distribution around implant bone interface. He has chosen the diameter and length as input and von-mises stresses as output. He has explained that length of the implant was more crucial in reducing stress and enhancing the stability of implants that diameter.

III. MATERIALS AND MODELS

Different perspectives regarding models and materials of the implants were taken by the various researchers in their study as follows-

Mohamed I. El-Anwar[1] in this research he has considered model of implants fixed to an edentulous mandible along with two co-axial cylinders [5]. The inner one represents the spongy bone that filling the internal space of the other cylinder that represents cortical bone. He has taken twenty five implants for research purpose having diameter range from 3.5 to 6 mm and length range from 9 to 13 mm. All implants have triangular thread patterns on them. The solid modeling and finite element analysis were performed on a personal computer Intel Pentium Core to Due, processor 3.1 GHz, 4.0 GB RAM.

Jenny Zwei-Chieng Chang [2] in this study, he has taken 4 types of mini-implants with bone structure and generated 3D models by using computer-aided design software (version 2005; Solid-Works, Concord, Mass). Here material for implants and bone assumed to be homogeneous, isotropic, and linearly elastic.

Liang Kong[3] in his research he selected a mandible segment with a screwed implant and a modeled was made on personal computer, using a 3D program (Pro/E Wildfire, Parametric Technology Corporation, USA). Here a cross-section of a mandible in the first premolar region was selected as prime data and that used for make solid model. This image was then extruded to create a three-dimensional mandible segment, which contained a thick layer of cortical bone surrounding the dense cancellous bone[9]. Materials of the model were considered to be isotropic, homogeneous and linearly elastic in nature. The implant was considered to be anchored in bone model along with entire interface.

T. Li [4] in his study he has selected maxillary posterior section with a screwed implant and a superstructure was modeled on a personal computer with a 3D program (Pro/E Wildfire, Parametric Technology Corporation, USA) [11][12]. He has used 9 samples of Strauumann implants along with the high solid abutment. He also has applied a porcelain superstructure with 2-mm occlusal thickness was applied over the titanium abutment. All materials used in the modeling were assumed to be isotropic, homogeneous, and linearly elastic [13][14].

IV. FEA METHODOLOGY

Mohamed I. El-Anwar[1] here he performed FEA analysis on each implant by using ANSYS version 9.0 and the used element in meshing all three-dimensional models is 8 nodes[7]. All models was analyzed under static conditions and they were subjected with four different loading conditions such as: tension of 50 N, compression of 100 N, bending of 20 N and torque of 2 Nm [6]. Loading was applied on the top middle node of each implant assembly in the studied models. Torque was generated by using two equal forces in magnitude, opposite in direction, applied to two opposite points on the diameter of implant head.

Jenny Zwei-Chieng Chang[2] has conducted FEA analysis of the 3D models of mini-implants by imported into MSC.Patran 2005 (MSC Software, Santa Ana, Calif) to generate a triangulated shell element mesh, and then transferred to MSC.Marc/Mentat2007r1 (MSC Software) for further finite element analyses. He has considered axial and lateral loading conditions to calculate stress around the implant and bone interface and evaluated the final results in terms of von-mises stresses. The total number of elements in the models ranged from 135,462 to 137,968, and the total number of nodes ranged from 12,916 to 13,064 with no degrees of freedom. He has evaluated the effect of thread depth at 0.16, 0.24, 0.32, 0.40, and 0.48 mm; and also investigated the influence of taper, the thread depth was fixed at 0.40 mm, and the core diameter of the 3 uppermost threads was gradually increased to create tapering of 0°, 3°, 5°, 7°, and 11°. To inspect the influence of taper length, the thread depth was fixed at 0.4 mm, and a 7° taper was set to the core of the 0°, 3°, and 5 uppermost threads, then he was validate the FEA result by conducting mechanical test on the same types of mini-implants.

Liang Kong[3] was used three dimensional finite elemental analysis approach to find the optimal range of thread pitch for which the stress distribution minimizes around the dental implant. The model was meshed and analyzed by ANSYS Workbench10.0 (SAS IP, Inc., USA). Here Models were meshed with 10-node-tetrahedron and 20-nodehexahedron. Elements. Models were composed of 170,000 elements and 250,000 nodes in average. The models were constrained in all directions and forces of 200 N and 100 N were applied axially.
and buccolingually respectively to the midpoint at the centre of a model [10]. Output was given in terms of max von-mises stresses in jaw bones and implants with respect to different pitch variations.

T. Li [4] has used FEA analysis for simulating the dental implants models which has many advantages over other simulating methods. All the models were meshed and analyzed by Ansys Workbench10.0 (SAS IP, Inc., USA). here under his study, implant was studied under screwed rough surface [10]. The models were meshed with 10-node-tetrahedron and 20-node-hexahedron elements. On average, one model consisted of 310,000 elements and 450,000 nodes and they were constrained in all directions. The loading conditions were applied under the 100 N and 30 N axial load. Here max EQV stresses and max displacement of implant-abutment complex were examined under given loading conditions.

V. RESULTS

Mohamed I. El-Anwar[1] was found that length, diameter and side area has the effect on stresses generated around the dental implants. For small diameter implants, it was observed that increment in length and side area reduced the stresses in cortical bone. For large diameter implants, increase in diameter was gave better stability and better performance.

Jenny Zwei-Chieng Chang [2] was conducted study on mini-implants design parameters such as depth, taper length and taper angle. He was observed that higher depth, short length and small taper gave maximum stresses around the bone–implant interface. Mini-implants also had relative larger displacements. He also found that pullout resistance was increased as thread depth increased from 0.16 to 0.32mm and after that it decreased considerably. Pullout resistance also decreased as taper angle and taper length has decreased. He also observed that maximum stresses were distributed on the uppermost threads at the neck of the mini-implants.

Liang Kong [3] was conducted experiment on five samples of the dental implants and he has chosen the pitch range from 0.5to1.6 mm and found impact of their variations on the equivalent stresses in cortical bone, cancellous bone and implant–abutment complex. His experimental Results showed that under axial load condition, above stresses decreased by 6.7%, 55.2% and 22.3% and under buccolinguinal load conditions, 2.7%, 22.4% and 13.0%, respectively with the variation of thread pitch. When thread pitch exceeded 0.8 mm, minimum stresses were obtained.

T. Li [4] has studied impact of implant length and diameter variation on stresses around implant and displacement of implant abutment complex. He has taken the diameter and length of the implants ranged from 3 to 5 mm and 6 to 14 mm respectively. His experimental results showed that under axial load conditions maximum von-mises stresses in cortical and cancellous bone decreased by 50% and 27% respectively and under buccolinguinal load by 52% and 60% respectively. Under the similar conditions maximum implant abutment displacement complex decreased by 39% and 43% respectively.

VI. DISCUSSION

Mohamed I. El-Anwar[1] in his study he has discussed about the different design aspects of the implants and their impact on stress distribution in cortical and spongy bones. He also explained that the increased diameter and length would give better stress distribution and increment in side area would also gave reduction in stresses on surrounding bone.

Jenny Zwei-Chieng Chang [2] has discussed about evaluated the mechanical characteristics of various mini-implant designs and studied the biomechanical outcomes of various mini-implant systems such as insertion torque, pullout strength, stiffness, and screw displacement before failure with the influence of the various design factors.[16][17] He was considered design parameters such as thread depth, degree of taper, and taper length. He also discussed greater thread depths, smaller taper degrees, and shorter taper lengths generated higher maximum stresses on the bone and thread elements and also had larger relative displacements. He was successfully conducted the experiment using FEA and those results were compatible with the actual tests done with the help of custom-made driving torque tester performed on implants inserted in synthetic bone blocks.

Liang Kong [3] in his study discussed that influence of the thread pitch on the maximum stresses generated in corical cancellous and implant abutment complex. His result explained that cancellous bone was more sensitive to the pitch variation than the cortical bone and under axial loading conditions pitch was played a great role than the buccolinguinal load.

T. Li [4] in his research he explained that implant success rate was depends on Bone quality and quantity (endogenous factors) and implant diameter and length (exogenous factors) [15]. He explained that the implant diameter and length were the key elements for the implant success because those factors were directly impact on primary stability, placement and removal torque values. In his study he discussed that length of the implant was more significant factor than the implant diameter on bone stresses in type IV bone.

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. Implant diameter and length are the most important key parameters for reducing the stresses around the implant-jaw interface.
2. Design parameters are also having important effect on mechanical properties of the implants.
3. We can conclude that thread configuration is one of the major contributors to initial implant stability. By Increasing the pitch of the thread, number of threads are been reduced but that hampers badly on the initial stability of implant.
4. Taper angle, taper lengths are also having significant impact on the primary stability and possibly reduce anchorage failures. Smaller taper degrees and shorter taper lengths generated higher maximum stresses on the
bone and thread elements and also had larger relative displacements.

5. Thread design, thread pitch, thread depth are also the considerable factors for improving the better performance of the implants and also reduces the generated stresses.

6. 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 spongy bone.

REFERENCES


About Authors:

Durvesh Jitendra Thakur is a Post Graduate student of Manufacturing system Engineering in Sarswati College Of Engineering Kharghar, Navi-Mumbai under Mumbai University, Maharashtra, India.

Prof. T. Z. Quazi is working as associate professor and HOD of Automobile engineering department in Sarswati College of Engineering, Kharghar, Navi-Mumbai. His work experience spans more than 17 years in teaching field.

Pankaj, N. Dhatrak is working as an Assistant Professor in Mechanical Engineering Department of MAER’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)