Dental Research Journal

Original Article

The influence of thread geometry on biomechanical load transfer to bone: A finite element analysis comparing two implant thread designs

Manisha G Herekar, Viraj N Patil, Shahnawaz S Mulani, Megha Sethi, Omkar Padhye

Department of Prosthodontics Crown and Bridge, Implantology, Maratha Mandals Institute of Dental Sciences and Research Center, Belgaum, Karnataka, India

ABSTRACT

Background: The success of dental implants depends on the manner in which stresses are transferred to the surrounding bone. An important consideration is to design an implant with a geometry that will minimize the peak bone stresses caused by standard loading. The aim of this study was to assess the influence of implant thread geometry on biomechanical load transfer and to compare the difference between two different thread designs.

Materials and Methods: A three-dimensional finite element model of D2 bone representing mandibular premolar region was constructed. Two implants of differing thread geometries, 13-mm length, and 4-mm diameter along with superstructures were simulated. One design featured fourfold microthread of 0.4-mm pitch, 0.25-mm depth in the crestal one-third; 0.8-mm pitch, 0.5-mm depth in the apical two-third. The other design had a single-pitch microthread of 0.8-mm pitch, 0.25-mm depth in the crestal one-third; 0.8-mm pitch, 0.5-mm depth in the apical two-third. A static axial load of 100-N was applied to the occlusal surface of the prosthesis. ANSYS CLASSIC 9.0 (PA, USA) software was used for stress analysis as von Mises stresses.

Results: A comparison of von Mises stresses between two thread designs revealed that fourfold microthread allows better stress distribution within the implant body by 43.85%, abutment by 15.68%, its superstructure by 39.70% and 36.30% within cancellous bone as compared to single-pitch microthread. The effective stress transfer to the cortical bone is lowered by 60.47% with single-pitch microthread.

Conclusion: Single-pitch microthread dissipates lesser stresses to cortical bone while the implant body, abutment, and superstructure absorb more stress. This will have a positive influence on the bone-implant contact and contribute to preservation of crestal bone. Implant with single pitch microthread will thus be preferable to be used in areas where the amount of cortical bone available is less.

Key Words: Implant, implant design, microthreads, thread design, thread pitch

Received: November 2012
Accepted: August 2013

Address for correspondence:
Dr. Shahnawaz S Mulani,
Department of
Prosthodontics Crown
and Bridge, Implantology,
Maratha Mandals Institute of
Dental Sciences and Research
Center, Bauxite Road, Near
KSRP ground, Belgaum,
Karnataka - 590 010, India.
E-mail: snawazzm@gmail.com

INTRODUCTION

The success of dental implants depends on the manner in which stresses are transferred to the surrounding bone. 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. Implant design features are one of the most fundamental elements that have an effect on implant primary stability and ability of implant to sustain loads during or after osseointegration.[1]

Access this article online

Website: http//:drj.mui.ac.ir

Implant design can be divided into two major categories: Macrodesign and microdesign. Macrodesign includes thread, body shape and thread design [e.g., thread geometry, face angle, thread pitch, thread depth (height), thickness (width) or thread helix angle]. Microdesign constitutes implant materials, surface morphology and surface coating.[2,3]

Biomechanical load management is dependent on two factors; the character of applied force and functional surface area over which the load is dissipated.[1]

Researchers have emphasized the importance of improving the functional surface area to maximize bone implant contact and improve stress distribution to the bone. From a bioengineering perspective, an important consideration is to design the implant with a geometry that will minimize the peak bone stress caused by standard loading. The complex geometry of the implants prevents the use of closed-form solutions in stress-analysis, where simple formulas relate the effect of external loads to internal stresses and deformations. Therefore, it is essential to understand stress concentration on implants that is affected by shape of thread end, pitch, the width of thread end, the height of thread, implant diameter and angle of inclination of implant.

Finite element analysis (FEA) is a technique for obtaining a solution to a complex mechanical problem by dividing the problem domain into a collection of much smaller and simpler domains or elements in which the field variables can be interpolated with the use of shape functions. This tool has been adapted from the engineering arena to dental implant biomechanics.[4]

The close apposition of bone to the titanium implant is the essential feature that allows transmission of stresses from the implant to the bone without any appreciable relative motion or abrasion. The absence of an intermediate fibrotic layer allows stresses to be transmitted without any progressive change in the contact between the bone and implant. The titanium implant and the bone may be regarded as having a perfect fit with no stress in either material prior to loading.

The main purpose of this study was to assess the influence of implant thread geometry on biomechanical load transfer and to compare the difference between two different implant thread designs- a single-pitch microthread (SPM) and a four-fold microthread (FFM).

MATERIALS AND METHODS

Solid models of a mandibular segment, dental implants and a porcelain crown developed using computer aided design (CAD) were used to construct implant bone FE models. The posterior mandible (from distal to first premolar to mesial to first molar) was harvested from a dry human skull and frontal sections of computerized tomographic (CT) images were obtained (1-mm interval between images). From each CT image material boundaries were delineated by an in house imaging program. This program employed various thresholds in CT number and searched for maximum gradient values of the CT number which were used to detect the boundary pixels between different materials. A depth–first algorithm was used to find the nearest boundary pixels and construct the contour of each material. The co-ordinates of points forming the contour lines were then imported into the FE software ANSYS classic 9.0 (PA, USA) to generate a solid 3D model of the mandible. The implant with its superstructure was modelled using Computer Aided Three-dimensional Interactive Application (CATIA, Avions Marcel Dassault, France) software. ANSYS classic 9.0 software was used for stress analysis.

Bone implant interface

The FEA model assumed a state of optimal osseointegration, which means that the cortical and trabecular bone were assumed to be perfectly bonded to the implant.

Modeling

A three-dimensional FE model of mandibular section bone was constructed. D-2 type of bone (according to Lekholm and Zarb classification)[5] which is more commonly found bone density in mandibular posterior region, having spongy center surrounded by 2-mm cortical bone of 24-mm length and 16-mm width was modeled. The implant and abutment complex was taken with implant length of 13 mm and 4-mm diameter [Figures 1 and 2].

The FFM implant [Figure 3] had a tapered body with taper angle of 1.5°, V-shaped four-fold microthread in the crestal one-third of implant with 0.4-mm pitch and 0.25-mm depth. The corkscrew thread in the apical two-third had 0.8-mm pitch and 0.5-mm depth (OSSTEM GSIII^{TM}, KOREA).

The SPM fixture [Figure 3] had a tapered body with taper angle of 1.5°, V-shaped microthread (0.8-mm pitch and 0.25-mm depth) in crestal third and corkscrew thread (0.8-mm pitch and 0.5-mm depth) in the apical two-third (OSSTEM TSIII^{TM}, KOREA).

The implants and the abutments were made up of titanium alloy. Mandibular second premolar was modeled according to Wheelers with crown

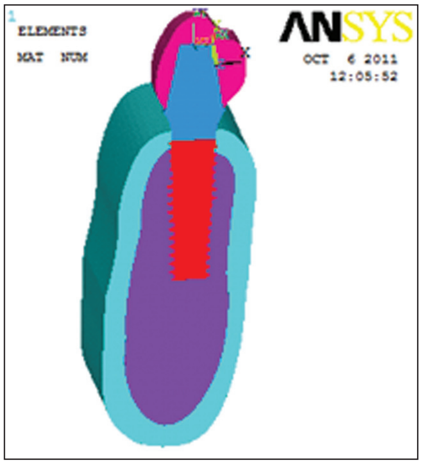

Figure 1: D-2 type of bone, having spongy center surrounded by 2-mm cortical bone

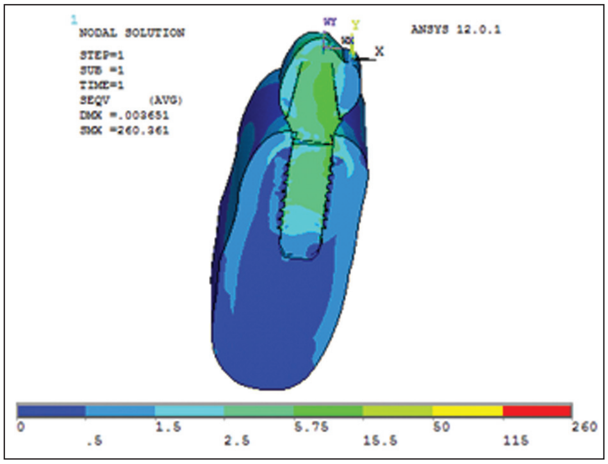

Figure 2: Completed meshed model of bone with implant, abutment, prosthesis framework and occlusal surface of prosthesis

Figure 3: OSSTEM GS III^{TM} and OSSTEM TS III^{TM} Implant models

framework of cobalt chromium alloy and occlusal surface of feldspathic porcelain. The metal and porcelain thickness was 2 mm. The model consisted of elements with nodes summarised below [Table 1].

Loading conditions

An average masticatory force of 100 N was determined from the literature. A 100-N static axial occlusal load was applied to the occlusal surface of crown in the central fossa to calculate the stress distribution. The stress levels were calculated as von Mises stresses.

Material properties

Porcelain and metal thickness used in this study was 2 mm. Cement thickness layer was ignored. All materials were presumed linear elastic, homogeneous, and isotropic. The corresponding elastic properties such as Young's modulus (E) and Poisson ratio (μ) were determined from the literature and are summarized below [Table 2].[6] A fixed bond between the bone and implant along the interface was selected which means that under the applied load on the implant, no motion between the bone and implant occurred. The final element on X-axis was assumed to be fixed and it defined the boundary condition.

Analysis and interpretation of results both numerically and by color-coding

The von Mises equivalent stress (MPa) at the implant-bone interface was computed using FEA software. All computations were performed on both the 3-D implant models and the values of maximum von Mises equivalent stress on the implant and the bone was obtained and were tabulated and analyzed for computation of the results.

Table 1: Material properties assigned to different material compounds of the finite element model

Material	Elastic modulus (E) (GPa)	Poisson ratio (μ)
Titanium (abutment, implant)	110	0.35
Spongy bone	1.37	0.3
Cortical bone	13.7	0.3
Co-Cr alloy (framework)	218	0.33
Feldspathic porcelain (occlusal material)	82.8	0.35

Table 2: Nodes and elements present in implant models

Model	Nodes	Elements
GS III	69626	307292
TS III	38577	119288

RESULTS

After application of static load of 100N on the central fossa of crown, von Mises stresses generated were calculated at the level of the cortical bone, cancellous bone, implant body, abutment, and occlusal surface of prosthesis.

High stress values were located at cervical cortical bone regions adjacent to implants in both models [Figures 4 and 5].

Comparison of stresses between two implants having different thread designs showed that the implant with fourfold microthread allows more stress distribution within surrounding cortical bone and decreased stresses to the cancellous bone, implant body, abutment, and its superstructure. Lesser stresses were noted at cervical cortical bone regions in the implant having the single pitch microthread [Table 3, Figure 6].

The results can be summarized as:

1. Stresses within cortical bone were lesser by 60.47% in the SPM implant.
2. Stresses within cancellous bone were lesser by 36.30% in the FFM implant.
3. Stresses absorbed by the implant body were lesser by 43.85% in the FFM implant.
4. Stresses absorbed by the abutment were lesser by 15.68% in the FFM implant.
5. Stresses absorbed by the prosthesis were lesser by 39.70% in the FFM implant.

DISCUSSION

Dental implants function to transfer load to surrounding biological tissues. The primary functional design objective is to dissipate and distribute biomechanical loads to optimize the implant supported prosthesis in function. A favorable implant design may compensate for risk for occlusal loads in excess of normal, poor bone densities, less than ideal implant position and number or less than an ideal implant size.[1]

Functional surface area (FSA) plays major role in addressing the variable initial Bone Implant Contact (BIC) zones related to bone density upon initial loading.[1] Screw-type implants have more functional surface area which is advantageous in softer bone types.[7]

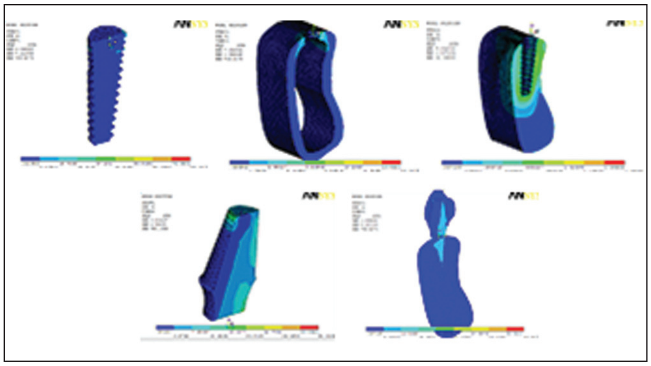

Figure 4: Stress values within OSSTEM™ TS III implant

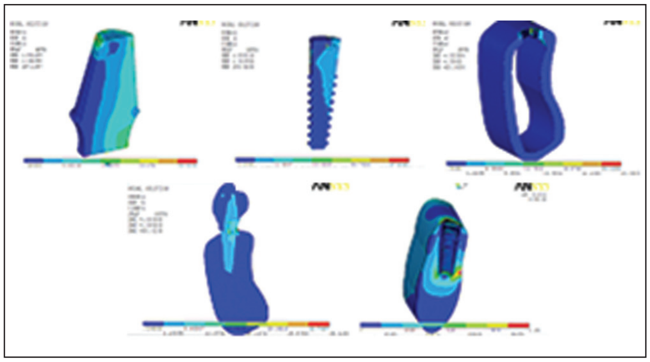

Figure 5: Stress values within OSSTEM™ GS III implant

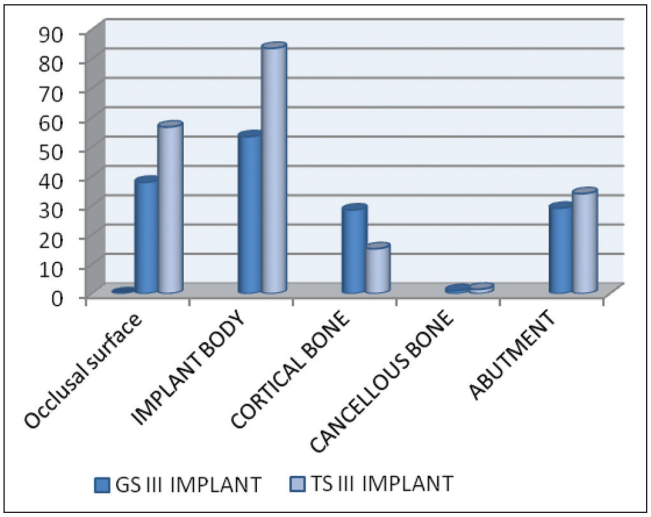

Figure 6: Comparison of stresses in von Mises between OSSTEM GS III and TS III implants

Threads are designed to maximize initial contact, enhance FSA, and facilitate dissipation of loads at the bone implant interface. Thread pitch is the distance from the center of the thread to the center of the next thread, measured parallel to the axis of a screw.[8] Smaller the pitch, more threads on the implant body for given unit length providing for greater functional surface area.

It has been shown that maximum effective stress decrease as screw pitch decrease and implant length increase.[9] However, it has also been reported that change of pitch has little influence on stress values.[10]

Table 3: Comparison of stresses in von Mises between OSSTEM GS III and TS III implants

	von Mises stress at variable sites (N/m²)				
Implant systems	**Occlusal surface of prosthesis**	**Implant body**	**Cortical bone**	**Cancellous bone**	**Abutment**
GS III Implant	38	53.54	28.58	1.06	29.16
TS III Implant	56.82	83.61	15.31	1.53	34.12

In the present study, thread shape and depth was same in both the models. Thread pitch was changed as 0.4 mm in FFM and 0.8 mm in SPM in crestal portion. Recently, the concept of microthreads in the crestal portion has been introduced to maintain marginal bone and soft tissues around the implants. Some authors attributed this bone loss to 'disuse atrophy'.[11] In presence of a smooth neck, negligible forces are transmitted to the marginal bone leading to its resorption. However, the presence of retentive elements at the implant neck will dissipate some forces leading to the maintenance of the crestal bone height according to Wolff's law.[12] Implants with microthreads in the coronal portion (81.8%) when compared with control non-microthreaded implants found increased BIC at 10 months.[13]

The jaw bone and implants are very complicated structures. Therefore, FEA can be utilized as a reliable tool to assess the stresses generated within various components that is not possible by *in vivo* studies. This study used the 3-D modeling technique as it has been proven that it is more accurate and valid than conventional 2-D modeling.

Type of stresses in finite element studies are generally described by means of direction (shear, tension, and compression) or by an effective absolute magnitude of principal stresses (equivalent stress of von Mises). The "equivalent stress of von Mises" is an expression that yields an effective absolute magnitude of stresses, taking into account principal stresses in three dimensions.[4]

Stress distribution in the FE model comes in numerical values and in color coding. Maximum and minimum value of von Mises stresses is denoted by red and blue color, respectively. The in-between values are represented by bluish green, green, greenish yellow, and yellowish red in the ascending order of stress distribution.

Results of this study indicate that FFM allows better stress distribution within the implant body by 43.85%, abutment by 15.68%, its superstructure by 39.70%, and 36.30% within cancellous bone as compared to SPM. The effective stress transfer to the cortical bone is lower by 60.47% with SPM. High stress values were located at cervical cortical bone regions adjacent to implants in both models.

FFM implant dissipates lesser stresses within the implant and the prosthesis as well as in the cancellous bone as compared to the SPM implant thus suggesting its favorable application in good quality bone. The SPM implant induces lesser stresses in cortical bone and generates more stress in implant. Thus, it would be prudent to suggest its applicability in softer bone types and areas prone to crestal bone loss like maxillary anteriors and especially in conditions with compromised bone quality or high occlusal stresses.

In this study, axially applied static loads have been assumed instead of more realistic dynamic-cyclic loads directed at occlusal angle encountered in the jawbone during mastication of food. Other assumptions made while modeling the implants also might have led to unrealistic results.[4]

CONCLUSION

Influence of microthreads has shown favorable results of stress distribution to the surrounding bone. SPM implant allows higher stresses to be absorbed by the implant body, abutment, and prosthesis and lesser stresses transferred to the cortical bone. This renders it favorable for all bone types and has particular significance in areas of poor bone densities as it would thereby help in preservation of available bone.

REFERENCES

1. Misch CE. Rationale for Dental implants. In: Misch CE, editor. Text book on Dental Implant Prosthetics. 3rd ed. St. Mosby: Louis; 2005. p. 1-229.
2. Geng MP, Ma QS, Xu W, Tan KB, Liu GR. Finite element analysis of four threadform configurations in a stepped screw implant. J Oral Rehabil 2004;31:233-9.
3. Geng JP, Xu DW, Tan KB, Liu GR. Finite element analysis of an osseointegrated stepped screw dental implant. J Oral Implantol 2004b;30:223-33.
4. Geng JP, Tan KB, Liu GR. Application of finite element analysis in implant dentistry: A review of the literature. J Prosthet Dent 2001;85:585-98.

5. Misch CE. Density of bone: Effect on surgical approach and healing. In: Misch CE, editor. Contemporary Implant Dentistry. 3rd ed. St. Louis Mosby: 2008. p. 645-67.
6. Eraslan O, Inan O. The effect of thread design on stress distribution in a solid screw implant: a 3D finite element analysis. Clin Oral Invest 2010;14:411-6.
7. Tada S, Stegaroiu R, Kitamura E, Miyakawa O, Kusakari H. Influence of implant design and bone quality on stress/straindistribution bone around implants: A 3-D finite element analysis. Int J Oral Maxillofac Implants 2003;18: 357-68.
8. Abuhussein H, Pagni G, Rebaudi A, Wang HL.. The effect of thread pattern upon implant osseointegration- review. Clin Oral Implant Res 2010;21:129-36.
9. Chun HJ, Cheong SY, Han JH, Heo SJ, Chung JP, Rhyu IC, *et al*. Evaluation of design parameters of osseointegrated dental implants using finite element analysis. J Oral Rehab 2002;29:565-74.
10. Liang D, Wang J, Ma Q, Lu Y. The influence of the screw thread and the height of constraints on the screw distribution around dent l implants by using three-dimensional finite element analysis. Shanghai Kou Qiang Yi Xue 2002;2:324-6.
11. Vaillancourt H, Pilliar RM, McCammond D. Finite element analysis of crestal bone loss around porous-coated dental implants. J Appl Biomater 1995;6:267-82.
12. Hansson S. The implant neck: smooth or provided with retention elements. A biomechanical approach. Clin Oral Implants Res 1999;10:394-405.
13. Abrahamsson I, Berglundh T. Tissue characteristics at microthreaded implants: An experimental study in dogs. Clin Implant Dent Relat Res 2006;8:107-13.

How to cite this article: Herekar MG, Patil VN, Mulani SS, Sethi M, Padhye O. The influence of thread geometry on biomechanical load transfer to bone: A finite element analysis comparing two implant thread designs. Dent Res J 2014;11:489-94.

Source of Support: Nil. **Conflict of Interest:** None declared.