Evaluation of design parameters of eight dental implant designs: A two-dimensional finite element analysis

SR Desai, MS Desai1, G Katti2, I Karthikeyan

Departments of Periodontology, H.K.E. Society’s S. Nijalingappa Institute of Dental Sciences and Research, 1Srivasa Dental Implant Center, Gulbarga, 2Pral Medicine and Radiology, Al-Badar Dental College, Gulbarga, Karnataka, India

Abstract

Aim: Implants could be considered predictable tools for replacing missing teeth or teeth that are irrational to treat. Implant macrodesign includes thread, body shape and thread design. Implant threads should be designed to maximize the delivery of optimal favorable stresses. The aim of this finite element model study was to determine stresses and strains in bone by using various dental implant thread designs.

Materials and Methods: A two-dimensional finite element model of an implant–bone system is developed by using Ansys. An oblique load of 100 N 45° to the vertical axis of implant as well as a vertical load was considered in the analyses. The study evaluated eight types of different thread designs to evaluate stresses and strains around the implants placed in D1 bone quality.

Results: Forty-five-degree oblique von Mises stresses and strains were the highest for the filleted and rounded square thread with an angulation of 30° (216.70 MPa and 0.0165, respectively) and the lowest for the trapezoidal thread (144.39 MPa and 0.0015, respectively).

Conclusions: The findings in this study suggest that the filleted and rounded square thread with an angulation of 30° showed highest stresses and strains at the implant–bone interface. The trapezoidal thread transmitted least amount of stresses and strains to the cortical bone than did other models.

Key words: Bone morphology, bone stress, dental implant

Date of Acceptance: 09-Nov-2011

Introduction

Implants could be considered predictable tools for replacing missing teeth or teeth that are irrational to treat.[1] Today, implant success is evaluated from the esthetic and mechanical perspectives. Both depend on the degree and integrity of the bond created between the implant and the surrounding bone. Many factors have been found to influence this interfacial bonding between the implant and the bone and thus the success of implants. Albrektsson et al.[2] reported factors such as surgical technique, host bed, implant design, implant surface, material biocompatibility, and loading conditions all have been shown to affect implant osseointegration.

Understanding these factors and applying them appropriately in the science of dental implants can lead us to achieve predictable osseointegration, thus minimizing potential implant failures. Finite element analysis (FEA) is, therefore, utilized in this work as an important tool to evaluate these effects and the implant biomechanical characteristics.
of different thread form configurations. It has also been widely used in the literature to evaluate the implant design and function and have predicted many design feature optimizations.[3–5] The FEA allows researchers to predict stress distribution in the contact area of implants with the cortical bone and around the apex of implants in the trabecular bone.

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 favorable 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.

Thread shapes that are available include V shape, square shape, buttress shape, and reverse buttress shape.[7] The original Branemark screw had a V-shaped threaded pattern.[8,9] While some manufacturers modified the basic V-shaped thread, others used a reverse buttress with a different thread pitch for better load distribution.[10,11] Knefel[12] investigated five different thread profiles and found the most favorable stress distribution to be demonstrated by an ‘asymmetric thread’, the profile of which varied along the length of an implant. Recently, it has been proposed that a square crest of the thread with a flank angle of 3° decreases the shear force and increases the compressive load (BioHorizons Maestro Implant Sysems Inc., Birmingham, AL).[13] 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. Thread configurations presently represented in the dental implant design include V-shaped thread (Nobel Biocare, 3I, Paragon, Lifecore), thin thread (IMTEC Sendax MDI), reverse buttress thread (Steri-Oss), and square thread (BioHorizons).

Reports have indicated that the biomechanical environment has a strong influence on the long-term maintenance of the interface between the implant and the bone.[14,15] A key factor for the success or failure of a dental implant is the manner in which stresses are transferred to the surrounding bone.[16] The interface can be easily compromised by high stress concentrations that are not dissipated through the implant configuration. It is necessary that biomechanical concepts and principles are applied to the thread design of the dental implant to further enhance the clinical success.

The objective of this study was to perform two-dimensional (2-D) finite element analyses on various shapes of the dental implant to find the optimal thread shape having more evenly stress distribution in the jaw bone.

**Material and Methods**

Computer-aided design (CAD) software was used to construct a model of the bone block based on a cross-sectional image of the human mandible in the molar region. The implant with a length of 10 mm and a width of 3.75 mm was constructed by using CAD software. After obtaining all the models, solid models were exported commercial FE software to generate the FE models. A 2-D finite element model of an implant–bone system is developed by using Ansys[Figures 1-5]. The use of the 2-D model is based on the fact that a proper 2-D model is much more efficient compared with its 3-D counterpart and the results can be as accurate, if only a qualitative study is required.[12] Plane strain analysis is used for structures.

**Load conditions**

A nodal force (load) is applied on the top of the transmucosal abutment (100 N) vertically. As the horizontal stress component of the engendered stresses induced by the nonaxial loading may affect the major remodeling in the interface between the bone and the implant significantly,[18] an oblique load of 45° to the vertical axis of the implant was considered in the analyses.

**Material properties**

The corresponding material properties are given in Table 1.[19] Finite element mesh

The finite element model was created by using the element topology: Plane 182 and global edge length=0.3 mm. The nodes over the free edges of the cortical bone were constrained in the x-, y- and z-directional rigid movement. The maximum node numbers used were 5737 and element numbers were 5562, and the number of nodes and elements for the models averaged a total of 2800 and 900, respectively. The shape of the mesh was quadrilateral. The length and width of the alveolar bone block was 22 and 20 mm. The length and width of the implant were 10 and 3.75 mm, the thread pitch was 0.8 mm, and the height of the thread was 0.4 mm for all the five models, respectively. Boundary conditions were of the support type and were applied at the nodes at the base of the mandibular model.

**Table 1: Properties of materials used in the analysis**

<table>
<thead>
<tr>
<th>Material properties</th>
<th>Modulus of elasticity (E) (GPa)</th>
<th>Poisson's ratio (ν)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Pure titanium</td>
<td>115</td>
<td>0.35</td>
</tr>
<tr>
<td>Compact bone</td>
<td>14.8</td>
<td>0.30</td>
</tr>
</tbody>
</table>
Implant shapes, based on various types of angulations of triangular thread implants to be selected for analyses, were as follows:
1. Triangular thread with an angulation of 47.5° (model 1a)
2. Triangular thread with an angulation of 55° (model 1b)
3. Triangular thread with an angulation of 60° (model 2a)
4. Triangular thread filleted at the base and flat at the tip with an angulation of 55° (model 2b)
5. Square thread with an angulation of 90° (model 3a)
6. Filleted and rounded square thread with an angulation of 30° (model 3b)
7. Trapezoidal thread (model 4a)
8. Buttress thread (model 4b)

Considering the boundary conditions, the bones were fixed horizontally and vertically in the x, y, and z directions at the base of the mandible model. The implant–bone interface was considered osseointegrated, as no contact pair was considered between the structures.
The jaw bone used in this study was assumed to be a homogeneous compact bone, i.e., D1 bone, because the major interest is to compare stress distribution of each model. The implants used were made of pure titanium. Both bone and implant were assumed to be homogeneous, isotropic, and linearly elastic. The thread of the implant was modeled as symmetric. The element sizes generated in the models were not identical. The downsized elements were used at the locations where the higher stress level was expected. The plane strain analysis is used for structures in which one dimension is much larger than the other two dimensions, and the cross section of interest is perpendicular to the long axis. This type of analysis is the best for a model of human mandible.

Results

Models 3b and 4a show, respectively, local stress distributions in region in which the highest stress occurred by different implant shapes in the jaw bone surrounding the dental implant with an oblique load of 45°. The maximum effective stress under loading condition occurred at the regions in the jaw bone adjacent to the first thread of implant. Forty-five-degree oblique von Mises stresses were highest (216.70 MPa) for filleted and rounded square thread with an angulation of 30° and lowest (144.39 MPa) for the trapezoidal thread [Figures 8 and 10]. Strains due 45° oblique load were highest (0.0165) for the filleted and rounded square thread with an angulation of 30° and lowest (0.0015) for the trapezoidal thread [Figures 7 and 11].

Discussion

The aim of this study was to find the pure effect on the bone stresses of variations of the thread shapes. For this reason it was assumed that all the parameters of the models were identical except the thread shape. This makes it possible to make a comparison between threads of different shapes.

Threads are used to maximize initial contact, improve initial stability, enlarge implant surface area, and favor dissipation of interfacial stress. Thread configuration is an important objective in the biomechanical optimization of dental implants. The interface can be easily compromised by high stress concentrations that are not dissipated through the implant configuration. It is necessary that biomechanical concepts and principles are applied to the thread design of the dental implant to further enhance the clinical success. FEA is, therefore, utilized in this work as an important tool to evaluate these effects, and the implant biomechanical characteristics of different thread form configurations. It has also been widely used in the literature to evaluate the implant design and function and have predicted many design feature optimizations. The FEA allows researchers to predict stress distribution in the contact area of implants with the cortical bone and around the apex of implants in the trabecular bone.

The finite element used was Plane 182. This element allows the analysis of a 3-D geometry. The element is defined by 10 nodes having three degrees of freedom at each node: Translation in the directions x, y, and z. These directions in the system of node coordinates correspond to the radial, axial, and tangential directions, respectively. Another advantage of the element Plane 182 is that it tolerates irregular forms without loss precision.

Element types are the triangular elements with two and three translational degrees of freedom at each node in the quadratic form. It is noted that a quadratic shape function provides a higher order interpolation of the displacement field, consequently more accurately modeling the stress and strain distributions, considering 0.8 mm as the optimal
Desai, et al.: Evaluation of peri implant stress in the bone thread pitch for achieving primary stability and optimum stress production on cylindrical implants with V-shaped threads.\[30\]

A key factor for the success or failure of a dental implant is the manner in which stresses are transferred to the surrounding bone.\[31\] Studies showed that a square thread design (as opposed to the standard V-shaped or buttress thread) was suggested to reduce the shear component of force by taking the axial load of the prosthesis and transferring a more axial load along the implant body to compress the bone\[32\] optimum for compressive load transmission as there is less shear load transmission than a V-shaped thread in a cylindrical implant.\[33,34\]

Misch et al.\[35\] suggested that V-shaped threads generate higher shear force than do square threads; the square threads generate the least shear force. Implants with V-shaped and buttress threads have been shown to generate forces that may lead to defect formation.\[36\] In square and buttress threads, the axial load of these implants is mostly dissipated through compressive force.\[37,38\] while V-shaped threaded implants transmit axial force through a combination of compressive, tensile, and shear forces.\[37\]

Our study evaluated eight types of different thread designs under 45° oblique loading condition to evaluate stresses and strains around the implants placed in D1 bone quality. Oblique loading has been used in the present study, which is suggested to represent a realistic occlusal load.\[39\] The study is designed to incorporate three triangular threads with different angulations and modification of one triangular thread having same angulation of 55°, which was filleted at the base with a flat tip, square and modified square, trapezoidal, and buttress threads.

Model 6 (filleted and rounded square thread with an angulation of 30°) showed highest stresses and strains compared to other models. A triangular thread having angulations of 55° (model 1b) was modified by making fillet at the base and flat at the tip (model 2b). There was a negligible difference in stresses and strains in between model 1b and model 2b, so the modification of the thread design was not advantageous enough to reduce stresses and strains.\[39\] The trapezoidal thread (model 4a) showed minimum stresses and strains compared to all other models. There was 33.36% reduction of von Mises stresses and 27.27% reduction of von Mises strains compared to model 6, which showed highest stresses and strains. Albrektsson et al.\[40\] recommended that the thread tops be rounded in order to relieve stress concentrations and predicted small stresses in the bone at interior points of the thread.\[40\] This is, however, a qualitative statement, and no recommendation was given as to the magnitude of the radius of curvature of the thread top, nor does the implant designer find any guidance concerning the values of flank angle, thread depth, pitch, etc., in the implant literature. There was a negligible difference of stresses and strains between other six models, and the values were in between the values of models 3b and 4a.

One of the limitations of this study is the simplified geometry of the bone model. Even though the strength of a bone block is similar to that of jaw bone, the strain patterns might vary with the bone geometry. In addition, the material properties of the FE maxillary model were assumed to be isotropic and homogeneous. The consideration of the anisotropic and inhomogeneous properties is still needed in future studies.
Conclusions

The preliminary results obtained by the present 2-D finite element model study suggest the following:

1. The filleted and rounded square thread with an angulation of 30° showed highest stresses and strains at the peri implant interface.
2. The square thread, the buttress thread, and the triangular thread with an angulation of 47.5°, 55°, and 60°, respectively, showed a similar amount of stress and strain in the bone.
3. The trapezoidal thread transmitted the least amount of stresses and strain to the cortical bone than did other models.

Furthermore, FEA and in vitro studies are needed with the simulation of D2, D3, and D4 bone qualities to evaluate and validate the results of the present study.

References