Load Transfer on Dental Implants and Surrounding Bones

Mohamed I. El-Anwar, Mohamed M. El-Zawahry and Mohamed El-Mofty

Mechanical Engineering Department, National Research Centre, Egypt
Prosthodontics Department, National Research Centre, Egypt
Faculty of Dentistry, Ain Shams University, Egypt

Abstract: The aim of this study is to study 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). Twenty-five different implant designs with gradual increase in diameter and length were placed in a cylindrical bone section and analyzed by Finite Element Method. Four types of loadings were applied on each design; tension of 50 N, compression of 100 N, bending of 20 N, and torque of 2 Nm, to derive design curves. Exact design equations and curves were obtained. The analysis of these results showed that, increasing implant diameter and length generates better stress distributions on spongy and cortical bones. Implant side area increase will reduce stresses on surrounding bones.

Key words: Dental Implant, Design, F.E.M., Prosthodontics

INTRODUCTION

Comprehensive study on microscopic phenomenon of the bone healing in 1952 was reported that the bone contacted on the titanium surface directly. This study led to animal study of endosseous implant. Human study was started in 1965 and he presented the results of 10 years of study (Branemark, 1977). In early development stage of dental implant, it had machined surface without any additional surface treatment. Although scientists have studied and developed the surface form and shape of implant, and achieving high success rate and predictable results over 40 years that has been utilized for several decades. Failed implants have been increased as compared with early development stage of implant (Esposito, et al. 2012). However there are many related factors affecting implant failure. Which may be the host related factors, such as patient age, gender, systemic disease, smoking and oral hygiene. or implant placement site related factors such as position in arch, bone quality and bone quantity. Also surgery related factors including an initial stability, angulation and direction of implant and the skillfulness of an operator. Implant fixture related factors, such as surface roughness, length, diameter, macrostructure and microstructure of an implant fixture. Implant prosthesis related factor that is prosthesis type, retention method (screw type or cement type), occlusal scheme and so on (Hee-Won Jang, et al., 2011).

A major parameter for the clinical success of endosseous implant therapy is the formation of a direct contact between implant and surrounding bone. The implant-bone response is thought to be influenced by implant surface topography. Some of the studies indicated a tendency for an increase of bone-to-implant contact with increasing roughness of the implant surface (Shalabi, et al. 2011). It is well known that the underlying biological phenomenon of osseointegration is thought to be a biological reaction cascade dividable into three distinct phases. The first and most important healing phase, osteoconduction, relies on the recruitment and migration of osteogenic cells to the implant surface. The second healing phase, de novo bone formation, results in a mineralized interfacial matrix equivalent to that seen in the cement line in natural bone tissue. These two healing phases, osteoconduction and de novo bone formation, result in contact osteogenesis, and given an appropriate implant surface, in bone bonding. The long-term remodeling of the tissue, the third healing phase, is influenced by different stimuli, the most important being the biomechanics of the developed healing site (John Davies, 2003). Bone structure around basal implants shows a dual mode of healing: Whereas direct contact areas show primary osteonal remodelling, in the void osteotomy-induced spaces, the repair starts with woven bone formation. This woven bone is later converted into osteonal bone (Stefan Ihdea, et al., 2008). Modeling of these tensions using graphics software for biomechanical analysis by three-dimensional finite element analysis is a promising alternative for this type of evaluation with the additional advantage of not being invasive and contributing for studies on hard-to-reach regions or impracticable conditions, such as measuring tensions, compressions and displacements in the implants and supporting structures (Gustavo, et al., 2009).

Generally, most stress analyses of dental implants using the finite element method (FEM) have been performed under static conditions. Despite the well-known static behavior of dental implants, a good understanding on the dynamic response of such dental implants is rather scarce (Yasuhiro Tanimoto, et al., 2009). The use of FEM in the mechanical analysis of dental implants has been described by many authors like Satoh, et al. 2005, Çaglar, et al. 2006, EL-Anwar 2009, 2011 and EL-Zawahry et al. 2009.
The aim of this study is to study 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). Twenty-five different implant designs with gradual increase in diameter and length were placed in a cylindrical bone section and analyzed by Finite Element Method.

MATERIAL AND METHOD

The understanding of dental implant mechanical behavior during loading process comes from clinical practice findings, where the problem of the durability prediction during implantation procedure arises. A set of finite element models describing possible different applied forces formed at implantation was chosen as a suitable method.

The geometric model generation in this study was based on previous works with the development of a model of implants fixed to an edentulous mandible (Cruz, 2001 and Cruz, et al., 2003). The major physiological and functional needs for root-form dental implants are demonstrated by the edentulous mandible, the badly resorbed edentulous maxilla or mandible without any anterior teeth (www.jada.ada.org).

Bone geometry was simplified and simulated as cylinder that consists of two co-axial cylinders. The inner one represents the spongy bone (diameter 14 mm & height 22 mm) that filling the internal space of the other cylinder (shell of 2 mm thickness) that represents cortical bone (diameter 16 mm & height 24 mm). The twenty-five different implant designs used in this study cover the diameter range from 3.5 to 6.0mm and length range from 9.0 to 13.0mm. EL-Anwar and EL-Zawahry 2011, simulated plain implant by a cylinder with removed part (0.5mm thickness). Triangular thread type was simulated in this study, where thread depth and pitch are of order 0.5mm. Each implant was subjected to four different loading conditions; tension of 50N, compression of 100N, bending of 20N and torque of 2Nm. The base of the finite element models set to be fixed which defined the boundary condition (Sevimay, et al., 2005). 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.

Linear static analysis was performed. The solid modeling and finite element analysis were performed on a personal computer Intel Pentium Core to Due, processor 3.1 GHz, 4.0GB RAM. The meshing software was ANSYS version 9.0 and the used element in meshing all three-dimensional models is 8 nodes Brick element (SOLID45), which has three degrees of freedom (translations in the global directions) (Kohnke, 1994). Mesh density is important parameter affecting solution accuracy, and analysis time. As the geometries are curved, improving the mesh has the usual effect of improving the results for the discrete model (increasing the obtained stress levels accuracy in regions of high stress gradients). Another effect of increasing the number of elements is to reduce sharp angles created artificially by the process of substituting the geometric model by the mesh, reducing artificial peak stresses by improving the representation of the actual geometry.

Number of nodes and elements of the twenty-five models with indication to the implant model geometry are presented in Table (1), while the material properties used in this study are listed in table 2.

Table 1: Twenty-five models (dimensions, number of nodes, and elements)

<table>
<thead>
<tr>
<th>Model</th>
<th>Diameter</th>
<th>Length</th>
<th>No of Nodes</th>
<th>No of Elements</th>
<th>Model</th>
<th>Diameter</th>
<th>Length</th>
<th>No of Nodes</th>
<th>No of Elements</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>3.5</td>
<td>9</td>
<td>18,086</td>
<td>104,680</td>
<td>16</td>
<td>5.0</td>
<td>9</td>
<td>16,335</td>
<td>94,629</td>
</tr>
<tr>
<td>2</td>
<td>10</td>
<td>9</td>
<td>19,650</td>
<td>114,035</td>
<td>17</td>
<td>10</td>
<td>18,213</td>
<td>105,980</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>11</td>
<td>11</td>
<td>21,927</td>
<td>127,677</td>
<td>18</td>
<td>11</td>
<td>21,075</td>
<td>123,190</td>
<td></td>
</tr>
<tr>
<td>4</td>
<td>12</td>
<td>12</td>
<td>24,692</td>
<td>144,358</td>
<td>19</td>
<td>12</td>
<td>23,313</td>
<td>136,778</td>
<td></td>
</tr>
<tr>
<td>5</td>
<td>3.5</td>
<td>13</td>
<td>26,077</td>
<td>152,658</td>
<td>20</td>
<td>5.0</td>
<td>13</td>
<td>26,445</td>
<td>155,686</td>
</tr>
<tr>
<td>6</td>
<td>4.0</td>
<td>9</td>
<td>17,721</td>
<td>102,801</td>
<td>21</td>
<td>6.0</td>
<td>9</td>
<td>14,704</td>
<td>84,996</td>
</tr>
<tr>
<td>7</td>
<td>10</td>
<td>9</td>
<td>21,766</td>
<td>127,086</td>
<td>22</td>
<td>10</td>
<td>15,851</td>
<td>91,862</td>
<td></td>
</tr>
<tr>
<td>8</td>
<td>11</td>
<td>11</td>
<td>20,968</td>
<td>122,169</td>
<td>23</td>
<td>11</td>
<td>18,215</td>
<td>106,057</td>
<td></td>
</tr>
<tr>
<td>9</td>
<td>12</td>
<td>12</td>
<td>23,596</td>
<td>137,999</td>
<td>24</td>
<td>12</td>
<td>19,272</td>
<td>112,535</td>
<td></td>
</tr>
<tr>
<td>10</td>
<td>4.0</td>
<td>13</td>
<td>24,981</td>
<td>146,357</td>
<td>25</td>
<td>6.0</td>
<td>13</td>
<td>22,184</td>
<td>130,093</td>
</tr>
<tr>
<td>11</td>
<td>4.5</td>
<td>9</td>
<td>17,094</td>
<td>99,293</td>
<td>12</td>
<td>10</td>
<td>20,146</td>
<td>117,801</td>
<td></td>
</tr>
<tr>
<td>12</td>
<td>10</td>
<td>11</td>
<td>21,290</td>
<td>124,474</td>
<td>13</td>
<td>11</td>
<td>21,090</td>
<td>124,474</td>
<td></td>
</tr>
<tr>
<td>13</td>
<td>12</td>
<td>12</td>
<td>23,704</td>
<td>139,101</td>
<td>14</td>
<td>12</td>
<td>24,953</td>
<td>146,581</td>
<td></td>
</tr>
</tbody>
</table>

Table 2: Material properties

<table>
<thead>
<tr>
<th>Material</th>
<th>Young’s Modulus [MPa]</th>
<th>Poisson’s ratio, ν</th>
</tr>
</thead>
<tbody>
<tr>
<td>Cortical Bone</td>
<td>1380</td>
<td>0.30</td>
</tr>
<tr>
<td>Spongy Bone</td>
<td>150</td>
<td>0.30</td>
</tr>
<tr>
<td>Implant (Titanium)</td>
<td>110,000 (Per ASTM E8-04)</td>
<td>0.35</td>
</tr>
</tbody>
</table>
Referring to previous publication by EL-Anwar and EL-Zawahry (2011), addition of triangular thread to the plain implants, increased the number of nodes and elements by 40 to 60%. In addition side area increased by 75 to 85%. For the same implant, increasing implant length among the specified range increases the side area up to 76.2%, while increasing implant diameter among the specified range resulted in increasing side area up to 71.4%. Such increase will affect stresses distribution among surrounding bones.

Analyzing F.E. results, by correlating generated stresses and deformations to implant length, diameter, cross sectional area to side area, can lead to important conclusions. Additionally, design curves were obtained from this study that may help in selecting suitable implant geometry to be used with the patient jaw-bone conditions and limit of stresses can be withstand.

**Results:**

Four runs on each of the twenty-five models were done simulating the four loading conditions prescribed for this study. Graphical comparisons are preferred in such huge number of models and load cases. While tabling and curve fitting using least squares method to obtain results can lead to design equations and curves.

Analyses of Figures 1 to 8, which summarize the results of this study, can guide to helpful and powerful design curves and equations. Linear trend lines (curve fitting) are fairly enough for extracting the required data. Increasing implant length for small implant diameters dramatically reduces the values of generated stresses on cortical bone. In other words increasing side area for implants with small diameters reduces the generated stresses on cortical bone. Side area can also be increased by adding threads and micro-threads, or changing thread type ... etc. In contrary, spongy bone received more energy by increasing implant length, which resulted in higher level of generated stresses.

In the present study; for small implant diameter (range 3.5 to 4.5mm) increasing implant length, dramatically decrease maximum tensile stress generated in cortical bone under tension loading (Figure 1). On the other hand, it increases in spongy bone (Figure 2). Large diameter showed more stable behavior under tension loading proving that implant diameter is the dominant factor not the implant length. Back to Figures 3 and 4 similar results can be found for small and large diameters.

Von Mises stresses in Figure 5 showed similar changes with increasing implant diameter under bending loading. While Figure 6 showed that Von Mises stresses under bending loading, appeared on spongy bone highly depend on implant diameter that reduces the ratio between side and cross sectional areas. Finally, Figures 7 and 8, showed the torsion loading results on cortical and spongy bones. Implants with large diameters are stable, while ones with small diameters can have better performance by increasing their length (side area).

![Graphical representation of results](image)

**Fig. 1:** Maximum tensile stress in cortical bone under 50N tension loading
Fig. 2: Maximum tensile in spongy bone under 50N tension loading

Fig. 3: Maximum Compression in cortical bone under 100N compression loading

Fig. 4: Maximum Compression in spongy bone under 100N compression loading
For short implants increasing diameter improves its effect on cortical bone (reduces the generated stresses). In these cases both side and cross sectional areas were increased. The ratio of side to cross sectional area is 4xLength/Diameter, i.e. increasing diameter reduces the ratio between side and cross sectional areas. Comparing stresses values on cortical bone while side area and cross sectional area increase, results showed more decrease of cortical bone stresses values by increasing side area.
Fig. 8: Stress intensity in spongy bone under 2Nm torque loading

As an example, Figure 9 is showing a sample of the obtained results of model no. 25 under compressive loading of 100N.

Fig. 9: sample of obtained results, model no. 25 under 100N compressive loading (stress intensity on implant, maximum compressive stress on cortical bone, and vertical displacement in spongy bone)

Discussion:
Different geometric configurations of implants were investigated in this study. Aiming to provide stresses analysis and to compare these implants biomechanical behavior. Even with the simplifications of considering the bone to be homogeneous and linearly elastic, symmetric muscle action, complete osseointegration and static load, the model results can be very close to the actual situations observed in clinical studies (Baiamonte, et al 1996, Geng, 2004, Ciftci, 2001, Geramy and Morgano, 2004 and Luigi Baggi, et al, 2008). Many of the options
adopted in the current model should be taken into account in the analysis of the results. Complete osseointegration is not observed in clinical studies, as the level of osseointegration is highly variable. In a 3D F.E. analysis of osseointegration percentages and patterns on implant-bone interfacial stresses, Papavasiliou and Colleagues (Papavasiliou, et al. 1997) concluded that different degrees of osseointegration do not affect the stress levels or distributions for axial or oblique loads. So, fixing a value of 100% in a comparative study does not affect the conclusions.

The consideration of a limit for the interface resistance between bone and implant, not included in the model, is an interesting topic to be included in future models, which requires a non-linear treatment of the problem considering contact and fracture at the implant bone boundary layer. In recent years, several studies had shown that a more precise consideration of the physical processes in F.E. models used in dental biomechanics can lead to reliable results (Çaglar, et al 2006). The modeling of whole mandible, the muscles, the temporomandibular joint and the supporting system, can bring the model more closer to reality (Cook, et al 1982). Three-dimensional modeling, special attention to boundary conditions, the use of a fine mesh with appropriate number of degrees of freedom, all these factors contribute to the precision of the computational results (Holmgren, et al 1998).

Accurate determination of the biomechanical scenario in the bone tissue surrounding implants is an experimental challenge, and these have been in the past performed by a variety of methods including photoelasticity, strain-gauge placement, and finite element analysis. Concerning finite element analysis, the ever evolving software and hardware capabilities provide researchers with new capabilities and tools in short periods of time (Luigi Canullo, et al. 2011).

To obtain information on the strength of the bone–implant interface, a mechanical evaluation of retrieved implants must be performed. For threaded implants, the most appropriate technique is torque testing. A tensile bench, which allows the application and maintenance of an exact reproducible perpendicular force on the implant (Junker, et al., 2010).

The cortical bone tissue had lower capacity to dissipate stress as well as a more uniform increase of the torque, showing higher principal maximum stress in comparison with the cancellous tissue. These results are similar to many studies which assessed the influence of the osteotomy diameter for implant placement and the stress concentration on implant threads. These results are explained by the different mechanical properties between cancellous and cortical bones. The computational analysis by finite elements shows great versatility in the analysis of complex models. This analytical method allowed identifying the homogeneity between different models with varied insertion torques that are difficult to obtain in an experimental study with physical models, as well as the same mechanical properties and dimensions for cancellous and cortical bones. The anisotropy found in the bone tissue was reproduced in the present study for the cortical and cancellous bones, being characterized by different stress responses under forces applied in varied directions (Bruno, et al., 2010).

When maximum stress concentration occurs in cortical bone, it is located in the contact area with the implant, and when the maximum stress concentration occurs in spongy (trabecular) bone, it occurs around the apex of the implant. In cortical bone, stress dissipation is restricted to the immediate area surrounding the implant; in trabecular bone, a fairly broader distant stress distribution occurs. Therefore, the percentage of bone contact is significantly greater in cortical bone than in trabecular bone. Cortical bone, having a higher modulus of elasticity than trabecular bone, is stronger and more resistant to deformation. For this reason, cortical bone will bear more load than trabecular bone in clinical situations (Matteo Danza, et al., 2009). A good understanding of implant biomechanics makes it possible to optimize the treatment plan for individual patients to reduce the risk of functional complications and failures. Clinically, it can be hypothesized that the suitable implant design providing the most equitable transfer of the occlusal forces among abutments are preferable from the standpoint of bone preservation (Asuka Haruta, et al., 2011).

Reactions between the surfaces of metallic materials and living tissues are the initial events that occur when the materials are implanted into the human body. Tissue compatibility is governed by the reactions in the initial stage. Thus the surface properties of materials are important (Takao Hanawa, 2011).

Misch (2007) states three functions for threads which are to maximize initial contact, enhance functional surface area and facilitate dissipations of stress at the interfacial area. So although the threaded forms had a higher peak of stress, the benefits of threads cannot be neglected. In evaluating the best form of thread for dental implants, three factors should be taken into account: thread shape, thread depth, and thread pitch. Misch (2007) relates all these factors to functional surface area per unit length of implant which is modified by these three parameters. The greater the thread pitch and depth, the greater the surface area if all other factors are the same. However, from the surgical ease point of view, the fewer the threads, the easier the bone tap or the insert of the implant.

The modeled muscular force action at bone surface generated stresses as high as those obtained around the implant, as shown in previous studies. This fact provides a qualitative way of comparing the obtained stress levels and suggests that modeling of the whole mandible is important and should not be neglected (Cook, et al 1982).
Conclusions:

Comparative F.E.M. stress analyses between different implant geometries or different implant prosthetic concepts under the same conditions have been previously reported (Holmgren, et al 1998 and Satoh, et al 2005). Comparisons under different modeling conditions could serve as a reference, but do not provide conclusive proof. However, different studies presented comparisons with the Bränemark system, used as a practical standard as it can provide predictable and thoroughly studied clinical results (Yokoyama, et al 2005).

Simulation results considered functioning implants, modeling crestal bone loss after a healing and loading period. These results have also highlighted the influence of implant length and diameter on load transfer mechanisms. In agreement with the numerically experienced trend proposed by (Himmlova, et al, 2004 and Bozkaya, et al, 2004) maximum implant diameter seems to affect stress peaks at the cortical bone but not at the trabecular region, whereas stress values and distribution at the cancellous bone implant interface are primarily influenced by implant length. Nevertheless, to control the risk of bone overload and to improve implant biomechanical stress-based performance, numerical results from the current study suggest that for short implants; implant diameter can be considered more effective design parameter than implant length. In this context, the results of this study can be considered to be complementary to similar, previously published studies (Himmlova et al, 2004 and Saab, et al 2007). Due to the simplified and different geometrical models usually used in these studies, (Himmlova et al, 2004, Bozkaya et al, 2004 Saab, et al 2007) quantitative comparisons cannot be made. Analogously, (Carter’s et al, 1996) hypotheses regarding the influence of the strain level of the bone on hypertrophic responses or bone resorption cannot be directly verified in a quantitative sense. The numerical simulations have confirmed that the risk of bone overload essentially affects regions around the implant neck (Luigi Baggi et al, 2008).

Numerical simulations results showed that implant design, in terms of both; implant diameter and length, crestal bone geometry, and placement site affect the mechanisms of load transmission.

Stress distribution pattern did not change from one implant to the other even with changing implant diameter or length. Values of stresses may alter a lot in case of changing implant diameter and/or length. Increasing implant length from 9 to 13 mm can improve its behavior by reducing stresses generated on both types of bones by 20 to 30%. On the other hand increasing the implant diameter from 3.5 to 6 mm reduces such stresses by 30 to 50%. Increasing implant diameter has dominant effect over increasing the implant length. Implant diameter increase affect implant cross sectional area by increase it up to three times, while increasing the implant length do not affect it.

The cross sectional area ratio to implant side area may alter the bone stresses or in other wards ratio between implant cross sectional area and side area can dramatically affect its surrounding bone stresses.

REFERENCES


