

Effect of Different Implant Length on Stress Distribution Around Dental Implants.



MEDICAL SCIENCE

KEYWORDS : 3-D finite element analysis, implant length, stress distribution

Dr. Shruti Anand

PG Student , Kothiwal Dental College and Research Centre, Moradabad

Dr. Samarth Kumar Agarwal

Professor, Department of Prosthodontics and Crown and Bridge, Kothiwal Dental College and Research Centre, Moradabad.

Dr. Swatantra Agarwal

Principal and Head of Department, Department of Prosthodontics and Crown and Bridge, Kothiwal Dental College and Research Centre, Moradabad

ABSTRACT

The purpose of this study was to evaluate the stresses distribution pattern at the implant bone and implant abutment interface with varying implant length. A 3-D edentulous maxillary model was generated from computed tomography (CT) images. Four implants were placed in the premaxilla and splinted with a titanium bar. Three different configurations were evaluated with the distal implant varying in length (8 mm, 10 mm and 13 mm). A vertical load (120 N) was applied on the titanium bar corresponding to the left second premolar region. Von Mises stress values were evaluated. The results showed that stresses at the periimplant bone as well as at the implant abutment junction decreased on both the sides as the implant length increased. The surface area contact between the bone and implant increases leading to the implant absorbing more of the load and thereby decreasing the stresses.

INTRODUCTION:

Development of an ideal substitute for missing teeth has been a major aim of dental practitioners for millennia. Endosseous dental implants are currently used to retain and/ or support prostheses for restoring completely or partially edentulous patients, for a variety of tooth loss scenarios.¹ Osseointegration is the mechanism by which an implant is biomechanically accepted by the jawbone. Primary stability is also important, as the bone is still in a state of repair and necessitates applied stresses that promote bone growth.² The interface must tolerate the occlusal forces without adverse tissue response. In natural teeth, the periodontal ligament acts as an intermediate cushion element to buffer the occlusal forces; however, in the osseointegrated dental implant, occlusal loads are transmitted directly to the surrounding bones. This could cause microfracture at the interface between the bone and implant, fracture of implant, loosening of components of the implant system, and unwanted bone resorption.

It is critical for the practitioner to fully grasp the relationship between various combinations of bone and implant parameters and the resulting stresses on the bone from a wide range of masticatory forces.²

Several implant concepts have been developed, and many implant types are commercially available in different sizes, shapes, materials, and surfaces. To analyze the effectiveness and reliability of endosseous implants, revealing possible risks of implant failure, stress analysis of bone implant mechanical interactions is important. The complex geometry of the coupled bone-implant biomechanical system prevents the use of closed-form approach for stress evaluation. Therefore, the behavior of endosteal dental implants can be investigated by using numerical techniques. Recently, the finite element method has been widely applied to prosthetic dentistry^{3, 4} to predict stress and strain distributions at periimplant regions, investigating the influences of implant and prosthesis designs,⁵ the magnitude and direction of loads,⁶ and bone mechanical properties,⁷ as well as modelling different clinical scenarios.⁸

The aim of this study was to evaluate and compare the stress distribution pattern with different implant lengths in maxillary implant supported prosthesis by 3 dimensional finite element analyses.

MATERIALS AND METHOD

Three dimensional finite element models of maxilla and the implants were constructed on a personal computer with Intel Core 2 Duo Processor, 4 GB RAM, 320 GB hard disk using a series of

computer software programmes - Materialise Interactive Medical Image Control System Version 8.11 (MIMICS), Hypermesh Version 10.0 and Analysis System Version 12.1 (ANSYS).

Material properties were taken from the literature. The Young's modulus (Mega Pascal) for titanium, cortical bone, trabecular (very soft) bone, trabecular (soft) bone, trabecular bone and intermediate bone were $1.17,000 \times 10^6$, $13,700 \times 10^6$, 200×10^6 , 700×10^6 , $1,370 \times 10^6$ and $5,000 \times 10^6$ respectively. The Poisson's ratio for titanium and bone were taken to be 0.33 and 0.30 respectively.

The model was divided into large number of elements and nodes. An overview of the meshes used in study and the number of elements and nodes for each model is given below. (Table I)

Table I: Number of Nodes and Elements for each model

Models	No. of nodes	No. of elements
Model 1	82951	444300
Model 2	88549	443938
Model 3	88597	443942

The bone was simulated to be of D2 density (dense to thick porous compact bone on the outside and coarse trabecular bone on the inside) in the anterior region and D3 (porous compact and fine trabecular bone) in the posterior region of the maxilla.

The implant model was generated using Computer Aided Reverse Engineering (CARE). CARE creates a computer model of an object through measurements of the tangible object, as it exists in the real world using a laser based range scanner. This model is metrically accurate within the acceptable limits.

The implant assembly consisted of Nobel Replace Tapered implants having internal tri-channel connection with a regular platform, 13 mm length, 4.3 mm diameter; and MultiUnit abutments of 4 mm length. The titanium bar used was 85 mm in length, 3 mm in thickness and 10 mm in width. The first step was measuring at the points along the surface of the implant, screw, the abutment and the titanium bar using vernier callipers. Each point has an x, y and z coordinate locating the point in 3 D space. The collection of these points is known as a point cloud. The point cloud and the detected features were then used by the CARE system to model the entire geometry of the implant, screw, abutment and the titanium bar. Implant models of lengths 8 mm, 10 mm and 11.5 mm were generated keeping the diameter of the implant constant.

Three configurations were generated in the present study. In each configuration, the implants of 13 mm length were placed vertically (perpendicular to occlusal plane) in the right canine region (B) and left canine region (C) and 11.5 mm length implant was placed in the right second premolar region (A). In all the configurations, the distal implant (D) length varied to 8 mm, 10 mm and 13 mm which were placed vertically with a straight abutment in the left second premolar region. (Figure 1)

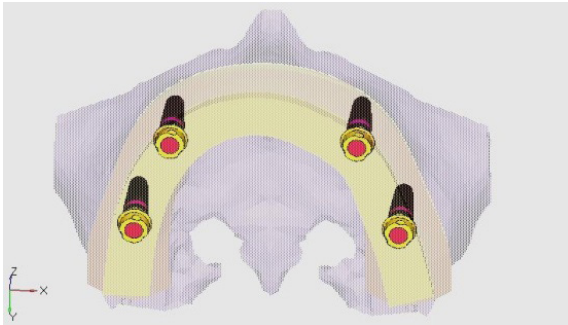


Figure 1: Configuration I designed with 4 cylindrical dental implants placed in the maxilla simulating the clinical treatment of edentulous patient with fixed complete denture.

A load of 120 N was applied on the titanium bar corresponding to the position of the left second premolar abutment. (Figure 2) A colour coded display of the pattern of von Mises stress at the implant abutment junction and the implant bone junction was made. The color coding used in the study depicted red as maximum and blue as minimum and the shades in between showed variation of stresses from maximum to minimum.

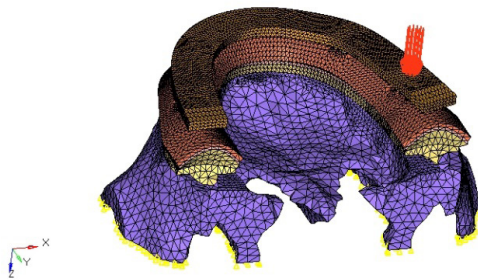


Figure 2: Load of 120 N applied vertically.

RESULTS:

The maximum stresses recorded at the implant bone and implant abutment interface with varying implant length are depicted in table II.

Table II: Stresses at implant bone interface and implant abutment junction of the left second premolar for all the configurations

	Stresses at Implant bone junction (MPa)	Stresses at Implant abutment interface (MPa)	Stresses at implant abutment interface (right second premolar region) (MPa)
Configuration I	5.821	8.261	0.280
Configuration II	5.358	8.131	0.273
Configuration III	4.897	7.754	0.248

When the four implants were splinted with a rigid fixed prosthesis, stresses at the periimplant bone and implant abutment

junction on both the sides decreased as the implant length increased. (Figures 3, 4, 5 and graph I, II, III)

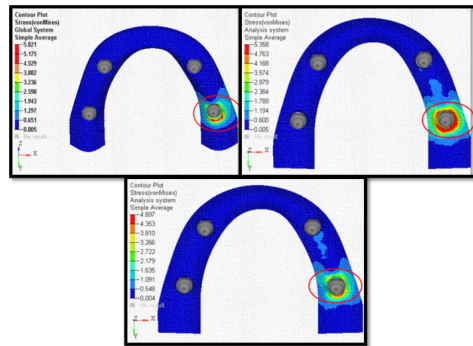


Figure 3: Simulation of stress transmission at periimplant bone in configuration I, II and III

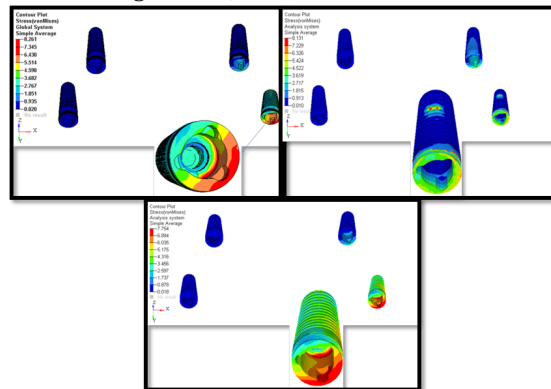


Figure 4: Simulation of stress transmission at implant abutment junction in configuration I, II and III

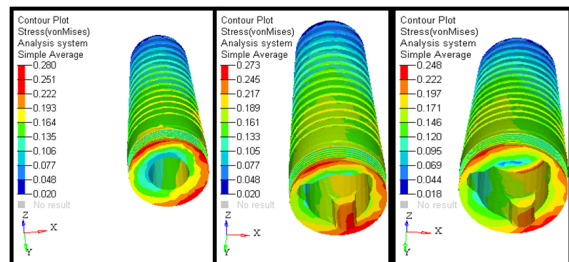
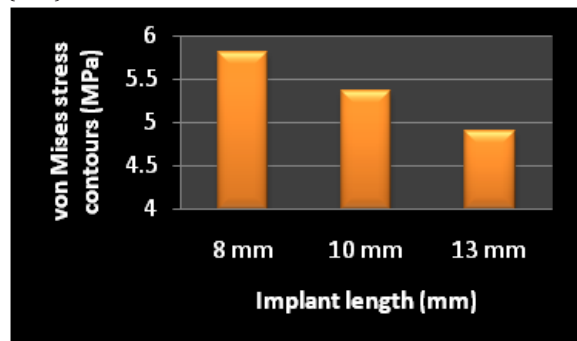
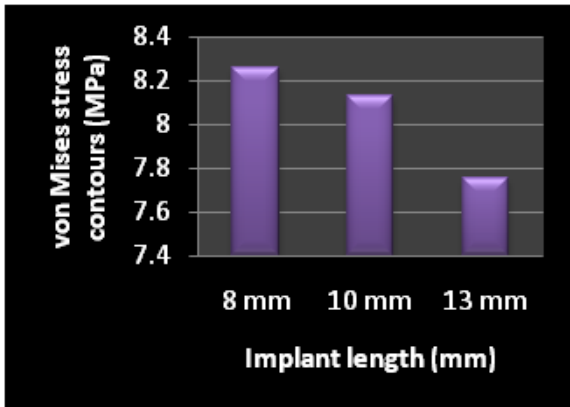


Figure 5: Simulation of stress transmission at implant abutment junction of right second premolar in configuration III

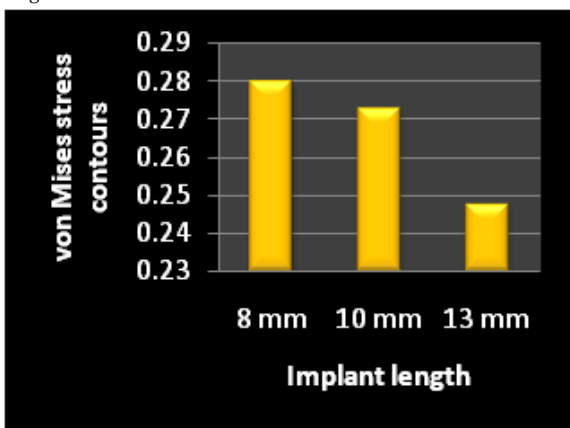
Graph 1: Von Mises Stress Contours at implant bone interface (MPa)



Graph 2: Von Mises Stress Contours at implant abutment interface (MPa)



Graph 3: Von Mises Stress Contours at implant abutment interface (MPa) of the contralateral implant with increasing implant length on the left side.



DISCUSSION:

The three dimensional (3-D) finite element models simulate with the nonsymmetric loading by the masticatory force on a dental implant results in a more satisfactory modeling of “clinical reality” than can be achieved. The results of the FEA computation depend on many individual factors, including material properties, boundary conditions, interface definition, and also on the overall approach to the model.

In the present study, stresses at the periimplant bone decreased as the implant length increased, indicating that stress distribution in the bone around the implant depends on the shape and the size of the implant. This is because the surface area contact between the bone and implant also increased, leading to the implant absorbing more of the load. Petrie et al⁴ in their study showing that increasing length caused as much as a 1.65-fold reduction in stresses. In a study conducted by Meijer⁹ it was found that in the model with connected abutments, the shorter the implant the more negative was the stress value; from -17.9 MPa with an implant of 13 mm to -23.2 MPa with an implant of 1 mm. Guan² found that the stresses decreased within the cancellous bone and cortical bone as the implant length increased.

CONCLUSION:

On the bases of present study the following conclusion can be drawn -

- 1) At the implant abutment junction also there was a decrease in the stresses with an increase in implant length.
- 2) Stresses were also decreased at the periimplant junction with an increase in implant length.

REFERENCE

1. Baggi L, Cappelloni I, Girolamo MD, Maceri F, Vairo G. The influence of implant diameter and length on stress distribution of osseointegrated implants related to crestal bone geometry: A three dimensional finite element analyses. *J Prosthet Dent* 2008;100:422-431. | 2. Guan H, Staden RV, Loo YC, Johnson N, Ivanovski S, Meredith N. Influence of bone on dental implant parameters on stress distribution in the mandible: A finite element study. *Int J Oral Maxillofac Implants* 2009;24:866-876. | 3. Geng IP, Tan KB, Liu GR. Application of finite element analysis in implant dentistry: a review of the literature. *J Prosthet Dent* 2001;85:585-98. | 4. Van Staden RC, Guan H, Loo YC. Application of the finite element method in dental implant research. *Comput Methods Biomech Biomed Engin* 2006;9:257-270. | 5. Petrie CS, Williams JL. Comparative evaluation of implant designs: influence of diameter, length, and taper on strains in the alveolar crest. A three-dimensional finite element analysis. *Clin Oral Implants Res* 2005;16:486-494. | 6. Holmgren EP, Seckinger RJ, Kilgren LM. Evaluating parameters of osseointegrated dental implants using finite element analysis: a two-dimensional comparative study examining the effects of implant diameter, implant shape, and load direction. *J Oral Implantol* 1998;24:80-88. | 7. Lin CL, Kuo YC, Lin TS. Effects of dental implant length and bone quality on biomechanical responses in bone around implants: a 3-D non-linear finite element analysis. *Biomed Eng Appl Basis Comm* 2005;17:44-49. | 8. Yokoyama S, Wakabayashi N, Shiota M, Ohyama T. The influence of implant location and length on stress distribution for three-unit implant-supported posterior cantilever fixed partial dentures. *J Prosthet Dent* 2004;91:234-240. | 9. Meijer HJ, Kuiper JH, Starmans FJ. Stress distribution around dental implants: influence of superstructure, length of implants, and height of mandible. *J Prosthet Dent* 1992;68:96-102. |