The Influence of Location of Occlusal Loading on Stresses Transferred to Implant-Supported Prostheses and Supporting Bone: A Three Dimensional Finite Element Study

Madhura Mangalvedhekar¹, Manisha Herekar²

¹Assistant professor, Department of Prosthodontics, P.M.N.M. Dental college and Hospital, Bagalkot.
²Professor, Department of Prosthodontics, Maratha Mandal's N.G. Halgekar Institute of Dental Sciences and Research center, Belgaum

Abstract

Background: The purpose of this study was to investigate the effect of loading at 1 to 3 different locations on the occlusal surface of a tooth on the stress distributions in an implant-supported mandibular fixed partial denture (FPD) and surrounding bone, using 3-dimensional finite element analysis. Material and methods: A 3-dimensional finite element model of a mandibular section of bone (Type 2) having spongy centre surrounded by 2mm cortical bone was modelled with missing second premolar and its superstructures. A two-stage 4x10 mm screw-shaped double tapered GS III (Osstem) dental implant was selected. Cobalt-Chromium was used as the crown framework material and feldspathic porcelain was used for occlusal surface. The implant and its superstructure were simulated in Altair Hypermesh v8.0SR1 program. Total loads at 300 N were applied at the following locations: 1) tip of buccal cusp (300 N); 2) tip of buccal cusp (150 N) and distal fossa (150 N); or 3) tip of buccal cusp (100 N), and mesial fossa (100 N). Results. The results demonstrated that vertical loading at location 1 resulted in high stress values within the bone and implant. Stress levels were observed within the bone for loading at locations 2 and 3; the former created the most extreme stresses and the latter the most even stresses within the bone. With loading at locations 2 or 3, stresses were concentrated on the framework and occlusal surface of the FPD, and low stresses were distributed to the bone. Conclusion. For the loading conditions investigated, the optimal combination of vertical loading was found to be loading at locations 2 or 3 which decreased the stresses within the bone.

Key words: Occlusal load, Stress, von Misses stress, Finite element analysis.

Received: 12 Mar 2014	Accepted: 2 May 2014	Published: 30 Aug 2014

INTRODUCTION

Dental implants function to transfer load to surrounding biological tissues. Thus, the primary functional design objective is to manage (dissipate and distribute) biomechanical loads to optimize the implant-supported prosthesis function.[1] Since dental implants were introduced for rehabilitation of the completely edentulous patients in the late 1960s, an awareness and subsequent demand for this form of therapy has increased. The use of implants have revolutionized dental treatment modalities and provided excellent long-term results.[2] The criterion for evaluating success of implant principally depends on the forces directed from implant body to the bone and the ability of the bone to withstand these forces.

In natural dentition, the periodontal ligament has the capacity to absorb stress or allow for tooth movement, but the bone-implant interface seemingly has no capacity to allow movement of the implant.[3] Load transfer at the bone-implant interface depends on the nature of the bone-implant interface; the quality and quantity of surrounding bone; the implant geometry; length, diameter and shape; the implant surface texture; the forces in relation to the type, magnitude, direction; the location of occlusal loading; the material properties of the implant and prosthesis.[4]

Address for correspondence
Dr. Madhura Mangalvedhekar, MDS.
Assistant professor,
Department of Prosthodontics,
P.M.N.M. Dental college and Hospital, Bagalkot
Tel No: 08095091819.
E mail: madhura.mangalvedhekar@gmail.com

Occlusion can be critical for implant longevity because of the nature of the potential load created by tooth contacts and the impact on the attachment of the bone to the titanium implant.[3] The implant must be capable of withstanding occlusal forces and be able to transfer the occlusal stress to the adjacent bone in correct orientation and magnitude such that tissue viability is maintained in a physiological state.

If occlusal force exceeds the capacity of the interface to absorb stress, the implant will fail. The biomechanical factors that contribute to implant overload, such as bone type, cuspal inclination, horizontal offset and occlusal anatomy are superimposed on physiologic variations.[5-7] Vertical loads from mastication induce axial forces and bending movement and result in stress gradients in the implant as well as in the bone.[3,8] A key factor for the success or failure of a dental implant is the manner in which stresses are transferred to the surrounding bone.[4] Deflective or premature contacts in the intercuspal position may be responsible for excessive force development. Gibbs et al.[9] reported that the greatest forces during mastication are exerted in the intercuspal position. If this position is unstable, intolerably high forces could be exerted.

Stresses and strains generated have been evaluated by methods like photo-elasticity, strain gauge analysis and finite element analysis. The finite element method is capable of providing detailed quantitative data at any location within a mathematical model.[10] Finite element analysis has been productive in producing data on force distribution to the implant and bone under vertical, horizontal and oblique forces.[11,12] Two and three-dimensional finite element analysis have been widely used to evaluate the stresses on various dental implant systems and its surrounding bone.[13,14]
This study was intended to evaluate the effect of location of occlusal loading on the stress distribution to 1) the occlusal surface of the implant supported prosthesis, 2) the implant framework, and 3) the surrounding bone around the implant.

MATERIALS AND METHODS

A 3-D finite element model of a mandibular section of bone with missing second premolar and its superstructures were used in this study. A mandibular bone model was selected, simulating D-2 type bone, according to the classification system of Lekholm and Zarb. A bone block, 24.2 mm high and 16.3 mm wide, representing the section of the mandible in the second premolar region, was modeled. It consisted of a spongy center surrounded by 2 mm of cortical bone.

A two-stage 4 X 10 mm screw shaped double tapered dental implant system (solid implant) (GS III; Osstem, Seoul, Korea) was selected for this study. Cobalt-Chromium was used as crown framework material and feldspathic porcelain was used for the occlusal surface. The implant and its superstructure were simulated using finite element software (Altair Hypermesh v8.0SR1). Porcelain and metal thickness used in this study were 0.8 to 2 mm. Cement thickness layer was ignored. The implant was positioned in a modeled cortical and trabecular bone block. 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 Table I. A fixed bond layering effect and the specific implant and crown produce a multilayer complex structure involving a GS III, Osstem solid implant system. The final element on the X-axis for each design was assumed to be fixed which defined boundary condition. The geometry of the tooth model has been described by Wheeler. The applied forces were static. Stress levels were calculated using von Mises stresses values. Von Mises stresses are most commonly reported in FEA studies to summarize the overall stress state at a point. The analyses were performed using ANSYS software program. Boundary conditions, loading, and mathematical model were prepared with Altair Hypermesh. The outputs were transferred to an ANSYS program to display stress values and distributions.

RESULTS

Implant and abutment

Stresses were concentrated at the neck of the implant (Fig. II). Maximum stress values were: 110.42 MPa for loading at location 1; 57.92 MPa for loading at 2 locations; and 57.32 MPa for loading at 3 locations at the neck of the implant.

Occlusal surface

The maximum stresses on the occlusal surface were concentrated on the distal fossae and buccal cusp for loading at location 2 (432.46 MPa). The maximum stresses were located on distal fossae and mesial fossae (335 MPa) for loading at location 2 and on buccal cusp (573.39 MPa) for loading at location 1 (Fig. III).

Cortical and spongy bone

Maximum stresses were located within the cortical bone surrounding the implant (Fig. IV). There was no stress within the spongy bone. Maximum stress values within the cortical bone were 32.18 MPa for loading at location 1, 30.26 MPa for loading at location 2 and 18.37 MPa for loading at location 3.

DISCUSSION

The term “organic occlusion” suggests that the supporting cusps of the premolar and molar teeth occlude with the opposing fossa with no less than 3 locations in centric relation. This study was designed to simulate the tripod occlusal contacts of the supporting cusp of the premolar tooth loaded at 1, 2, or 3 locations.

To gain insight in the biomechanics of oral implants, it is crucial to obtain an integrated understanding of the bone behaviour around oral implants. Assuming tight contact between the components of the implant system, it is likely that most of the recorded forces will be transferred to the implant. The effect of the reported forces on stress magnitude and distribution in the bone surrounding the implants may be investigated by means of finite element analyses.

The finite element model created for this study was a multilayered complex structure involving a GS III, Osstem solid implant and a layered specific crown structure. The stress following loading may be influenced greatly by the materials and properties assigned to each layer. It is also important that the layering effect and the specific implant and crown produce a cumulative result at the implant-bone interface that is specific for this model.
The structures in the model were all assumed to be homogeneous and isotropic and to possess linear elasticity. Additionally, 100% implant-bone interface was established, which does not necessarily simulate clinical situations. Also, the stress distribution patterns simulated may be different depending on the materials and properties assigned to each layer of the model and the model used in the experiments. Thus, the inherent limitations in this study should be considered.

The design of the occlusal surface of the model may influence the stress distribution pattern. In the current study, the locations for the force application were specifically described as cusp tip, distal fossa, and mesial fossa. However, the geometric form of the tooth surface can produce a pattern of stress distribution that is specific for the modeled form. The pattern could be different with even moderate changes to the occlusal surface of the crown. This occlusal form used for this model would not be expected to be the same for all premolar teeth.

Williams et al.[16] investigated the effect of stresses on cantilevered prostheses attached to osseointegrated implants by finite element analysis and stated that Co-Cr alloy reduced the maximal and effective stresses. The much higher elastic modulus of Co-Cr allowed more uniform distribution of stress within the framework, providing more efficient and durable load transfer.

Cibirka et al.[17] in an in vitro simulated study, compared the forces transmitted to bone by gold, porcelain, and resin occlusal surfaces and found no statistically significant differences in the force absorption quotient of the occlusal surfaces among these 3 materials. Porcelain is a commonly used material for occlusal surfaces. Therefore, porcelain was used for occlusal surface in the current study.

In the present study, a 4 X 10 mm screw shaped GS III, Osstem dental implant was selected because no report on the standard 4 mm diameter solid screw implant could be found in the literature.

Prostheses supported by 1 or 2 implants replacing missing
posterior teeth are subjected to an increased risk of bending overload.[7] The type of loading may influence the stress patterns developed. The present study showed the stresses in the implant to be similar when the vertical loads were applied with 2 or 3 locations but higher when the vertical loads were applied with 1 location.

Hoshaw et al[8] reported that overloading of implants resulted in increased bone resorption around the implant collar and a decreased percentage of mineralized bone tissue in the cortex. Papavasiliou et al[23] investigated the effect of the degree of osseointegration on stress distribution and found higher crestal stresses than apical stresses under all conditions. Excessive dynamic loading may decrease bone density around the neck of implants and lead to crater-like defects. The results of this current study are in agreement with the findings of these researchers. The stresses were concentrated in the neck of implant due to the rigid connection between the implant and bone. The modulus of elasticity of cortical bone is higher than spongy bone, and for this reason, cortical bone is stronger and more resistant to deformation. A consistent observation from the current study was concentration of maximum stress at the bone-implant interface at the level of cortical bone.

The results of this study, using vertical loading at 2 or 3 locations, produced high stresses on the occlusal surface of the implant-supported fixed partial denture and low stresses distributed to the bone. These types of loads would appear to minimize force transmission to the implant and surrounding bone. Vertical loading at 1 location produced stresses on cortical bone and implant, while low stresses were distributed to the occlusal surface. This may influence the maintenance of osseointegration.

CONCLUSION

Within the limitations of the study, following conclusions were drawn: The maximum stresses on the occlusal surface were concentrated on the buccal cusp for loading at location 1 (573.39 MPa). Loading at location 1 induced higher von Mises stress values (110.42) within the implant than loading at location 2 and 3. Loading at location 1 induced higher von Mises stress values (87.68 MPa) within the bone than loading at location 2 and 3. Loading at locations 2 and 3 decreased von Mises stresses within the bone. Von Mises stresses were concentrated on the implant and occlusal surface of the fixed partial denture.

REFERENCES

