FINITE ELEMENT ANALYSIS OF A DENTAL IMPLANT SYSTEM WITH AN ELASTOMERIC STRESS BARRIER

Lídia Carvalho (1,2), António Ramos (1), José A. Simões (1)

(1) Departamento de Engenharia Mecânica
Universidade de Aveiro
3810-193 Aveiro
Portugal

(2) Instituto Superior de Saúde do Norte
4585-116 Gandra
Portugal

ABSTRACT

This paper describes a numerical study performed with the finite element method of novel dental implant system. A conventional Brånemark dental implant system was redesigned and a bio-inert stress barrier (elastomer) was interposed between the implant and the ceramic crown. The goal was to attenuate the loading of the bone surrounding the implant with high magnitude stresses. The new design was assessed and the equivalent von Mises interface stresses compared with the ones provoked by the conventional implant. Overall, the novel implant provoked lower interface stresses due to the stress shielding effect of the elastomeric stress barrier.

INTRODUCTION

The replacing success of a tooth relies on the mechanical and biological capacity of the anatomical substitute to replace lost physiological functions, mainly the masticator one. The clinical success of dental implants depends strongly on initial stability and long-term osseointegration that provides lasting incorporation into the bone media and depends on implant design features such as materials, geometry and fixation methods. The investigation trend, concerning dental implant design, is concerned with the development of anchoring design fixtures that can also participate in qualitative and quantitative adjustment of the continuing remodeling of interfacial bone. Therefore, future functional prostheses should take into consideration the osseoperception capacity of osseointegration [1].

The support of teeth and implants is inherently different. Tooth is viscoelastically supported in the bone, promoting an elastic deformation pattern, while the implant, due to its stiffness, is fairly much more rigid. It has been speculated and suggested that shock-absorbing material can be incorporated into prosthetic constructions to imitate the function of the viscoelastic periodontal membrane [1]. The purpose of the study was to assess the role of dental implant design with occlusion loads transferred to the surrounding bone media using the finite element method. The effect of occlusion loading aiming the minimization of the physiological deviation pattern was studied.

MATERIALS AND METHODS

The finite element method was used within the study. Part of a cadaveric mandible and implant (Nobel Biocare, Brånemark dental system, standard model of 3.75 mm diameter and 15 mm long) were modeled using SolidWorks® CAD software. Figure 1 illustrates the finite element mesh of the implanted mandible [2].

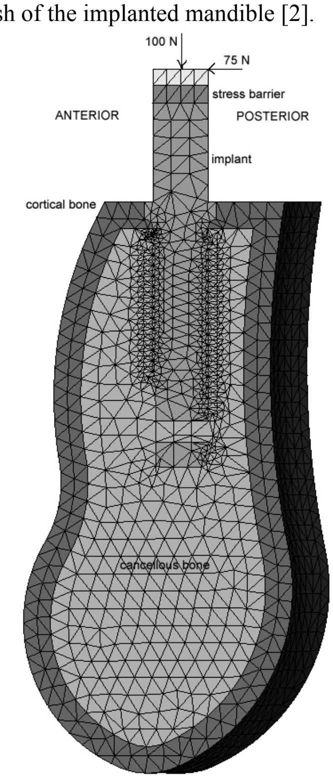

Figure 1. Finite element mesh of the implanted mandible

All structural materials, cortical and cancellous bone, implant and elastomeric stress absorber, were considered isotropic and with linear elastic behavior (elastic modulus of the implant, cortical bone, cancellous bone and elastomeric material equal to 110 GPa, 20 GPa, 9 GPa and 0.006 GPa respectively; Poisson's ratio equal to 0.25 for the implant, 0.33 for the bone structures and 0.49 for the elastomer).For the non conventional implant, a 0.5 mm thickness of an elastomeric material was added to the top of the surface of the implant. Two types of loading configurations were addressed: a purely compressive load and a load system provoking compressive and bending stresses. For the compressive load configuration, a load of 100 N was applied; for the bending configuration, a transverse force of 75 N was added, applying a bending moment. The finite element mesh was composed of 32025 nodes and 21285 linear tetrahedral elements. The lateral parts of the mandible were rigidly fixed.

RESULTS AND DISCUSSION

Figures 2 and 3 illustrate the von Mises stress distributions at the implant-bone interface for the compressive and bending loading configurations respectively. The stress values were obtained at the nodes in the inner surface of the thread of the implant. However, results were not significantly different at the nodes of the outer surface of the thread. Results of figure 2 show that the stress barrier reduces stresses at the anterior aspect of the interface. In fact, a 45% reduction was observed relatively to the results obtained for the conventional implant. For the bending configuration, the von Mises stresses were lower at the posterior aspect of the mandible.

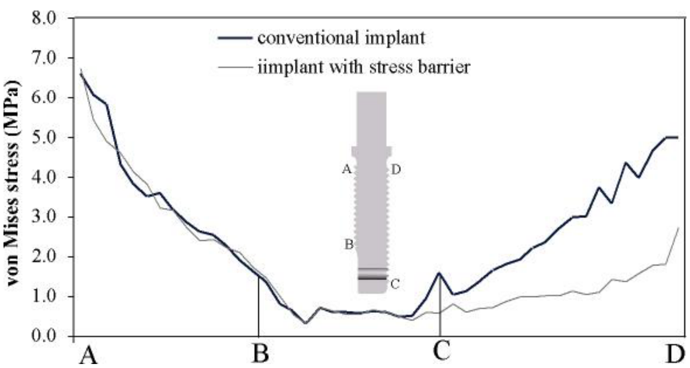

Figure 2. von Mises stress distribution (compression loading)

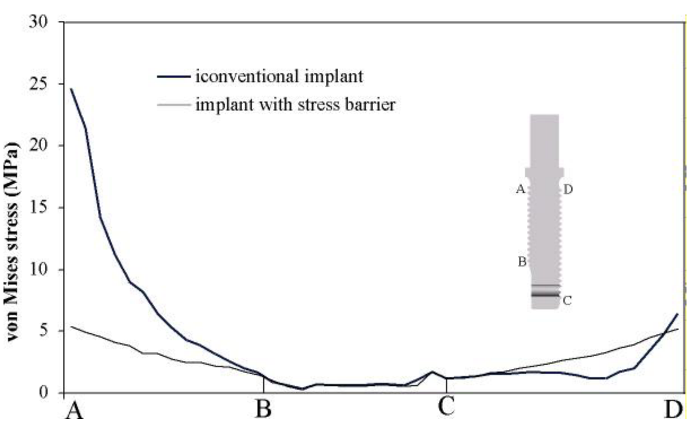

Figure 3. von Mises stress distribution (bending loading)

The thread of the implant was modeled following a helix curve, were a non-symmetric implant was modeled and therefore the non-symmetric von Mises stress distribution observed. However, the peak stresses at point A and E are not significantly different, 6.7 MPa and 5.0 MPa respectively.

Overall, the stress barrier avoids the transference of high stress to the surrounding bone. Due to the geometry of the implant, high stress concentrations are localized at the collar region. For the non conventional design the highest stress was observed at these regions, 16 MPa at the anterior and posterior aspects of the mandible. Stresses of magnitude 25 MPa and 9 MPa were observed at the posterior and anterior aspects for the conventional design.

An important aspect is related to the stresses developed in the elastomeric material. Equivalent von Mises stresses of the order of 2 to 3 MPa were obtained within the simulations preformed, which are bellowing the yield stress of the material used. An equivalent shear stress of 0.8 MPa was observed. However, at the metal-elastomer material interface, the stresses were significantly high, of the order of 17 to 20 MPa. Since the elastomer used in our study has a yield stress of 9 MPa, far bellow the interface stresses. This means that some problems can arise at the interface were the elastomer is placed. The maximum shear stress at this interface was 3.5 MPa. Further work has to be developed to critically localize the stress barrier within the implant to minimize interface stresses.

CONCLUSIONS

The study performed showed the importance of dental implant design on the occlusion load transfer mechanism. A stress barrier can avoid the transmission of high stress gradients, which can provoke the implant surrounding bone tissue fracture. These high stresses can also be responsible for abnormal bone remodeling. Threse stresses can be so high that even the bone remodeling process is not sufficient to avoid local trabecular damage.

The numerical model used within the study suffers from some important limitations, namely those related to the bone materials and elastomer characterization. Even though, the use of a shock-absorbable structural material seems to play an important key role within the load transfer mechanism and bone remodeling response. In vivo studies can clarify the advantageous on the use of a dental implant with this design characteristic.

AKNOWLEDGMENTS

The authors gratefully acknowledge the Fundação para a Ciência e a Tecnologia do Ministério da Ciência e do Ensino Superior for funding this work through project POCTI/42474/EME/2001 and PRODEP3, Acção 5.3 (co-financiada pelo Fundo Social Europeu) for funding Lídia Carvalho.

REFERENCES

1. Waters, N. E., 1980, "Some mechanical and physical properties of teeth", In Symposia of the Society for Experimental Biology, Vol.34 (The Mechanical Properties of Biological Materials), Cambridge University Press, London, pp. 99-135.
2. Carvalho, L., Pereira, J., Ramos, A., Simões, J. A., 2002, "Sistema de implante dentário não convencional: um estudo preliminar", Metodos Numericos en Ingenieria V, J. M. Goicolea, C. Mota Soares, M. Pastor e G. Bugeda (Eds.), Sociedade Española de Métodos Numéricos en Ingeniería, pp. 74.