



25th ABCM International Congress of Mechanical Engineering
October 20-25, 2019, Uberlândia, MG, Brazil

COB-2019-0403

NUMERICAL ANALYSIS OF INTERFACE OF DENTAL PROSTHESES THROUGH DAMAGES MECHANIC

Renan Archangelo dos Reis

José Aparecido Lopes Júnior

Universidade Tecnológica Federal do Paraná – Câmpus Cornélio Procópio

renanarchangelo@hotmail.com

junior_lopes1112@hotmail.com

Abstract. *Currently, in dentistry, after tooth loss for various reasons, these are being replaced by dental implants. The importance of biomechanical computational models is increasingly common, as these tools allow simulating the behavior of these devices that attempt to restore the functions of missing teeth. As the biomechanical aspects of implants are different from those of a natural tooth, surrounded by a periodontal ligament, the transfer of load to the implant and from the surrounding bone can generate efforts that, in addition to causing rehabilitation failures, may even exceed the physiological limit and cause loss of osseointegration. In the analysis of these prosthesis systems, it is essential to adequately represent the influences between the different implant / crown components, which are commonly joined by screws. Failure of these joints can impair the proper functioning of the prosthesis, or even produce unforeseen efforts, which are responsible for severe damage to the prosthesis or bone. In the present work, high aspect ratio tetrahedral solid finite elements are used to represent the interface surface between the implant / screw / crown contact components. A constitutive damage model is employed to reproduce the behavior of these interface elements. The model is developed to represent the differential behavior in traction and compression on the contact surface, allowing the separation of the components without offering practically resistance, but at the same time, preventing interpenetration movements in the case of compressive stresses on contact. Under these circumstances, it is expected that this methodology will be able to represent the behavioral aspects of the interfaces, presenting different structural behaviors, as well as their stress, when required for traction or compression*

Keywords: *Finite Elements Method, Damage Mechanic, Model, Interface, Biomechanics.*

1. INTRODUCTION

Many studies have been conducted in the area of biomechanics applied to prostheses proposing not only to improve the system, but also to improve patient comfort, avoiding failures and fractures in the prostheses, attenuate mechanical wear due to the stresses to which the structure is submitted and even accelerate the recovery process by developing new designs and utilizing other materials for this system.

With the emergence of various types of dental prostheses, doubts arise as to the mechanical behavior related to the possibilities of failure of the conjunct prosthesis / implant / bone, as well as the importance of understanding the behavior of parts and forces throughout the system and the effects that they can cause.

The correct adaptation of the crown / implant / retention screw assembly is biomechanically relevant for implant prostheses, since osseointegration is not resiliently performed on the alveolar bone (Weinberg, 1993). Therefore, passivity between prosthesis and implant is aimed at preventing inadequate stress concentrations from being generated between the components of this system (Millington; Leung, 1995; Duck et al., 2001; Kunavisarut et al., 2002), as well as its transmission to adjacent bone tissue (Skalak, 1983).

Passivity between the prosthesis and the implant is achieved when the retention screw is joining the structures only by a locking force, implying minimal bone stress in the absence of occlusal loading (Mulcahy et al., 2000).

For this, perfect adaptation between components and implant is necessary, as the presence of mismatches may prevent proper seating between parts during preload application due to the typical hardness of the prosthesis and retention screw (Patterson & Johns, 1992).), leading to asymmetric contact between the various components of the system (Isa & Hobkirk, 1995; & Hobkirk, 1996).

One of the major wear problems in engineering is the contact between two parts, not only because relative movement causes wear, but because the parts are in contact and under pressure, thus generating intense stresses that can cause interior fractures due to fatigue, which is subjected mainly to the chewing process.

The contact problem in dental prostheses has been poorly studied, and their emphasis is on stresses generated due to direct demands on the prosthesis and its contact with the bone rather than the stresses generated directly by the contact between prosthesis components that can often , causing high stress and lead to implant cracks.

The study of contact in dental prostheses is important because of the most common mechanical problems due to contact, but the analysis of the same is a nonlinear problem, which makes its analytical calculation difficult, so it is necessary to use computational numerical tools for resolution of this problem. For this, the finite element analysis (FEM) is a tool used in engineering.

In recent years, several constitutive models have been developed to simulate the effects of microstructural changes on the mechanical behavior of materials (Cervenka; Papanikolaou, 2008). The mechanic of continuous damage, according to Lemaitre (1996), is concerned with the solid load capacity without major cracks in which the material itself is damaged due to the presence of micro defects such as micro cracks and microvessels. Micro defects contribute to the post-peak nonlinear response, which becomes macroscopically evident by the reduction of material stiffness and resistance.

Constitutive damage models have been used as a tool to analyze structural stiffness loss to predict material degradation. Its interest is the simulation of the mechanical degradation of almost fragile materials, such as concrete, ceramics and rocks, which, after the elastic regime, the stress reduction occurs at each deformation increment, outlining the nonlinear behavior of the material. For the development of appropriate tools, it is indispensable that the nonlinear property of these materials be known and precisely modeled, especially their state of damage.

For the development of this work, it was used (to represent the interaction in contact between the prosthesis components) a constitutive model of combined damage, which is able to represent the differentiated behavior in traction and compression for almost fragile materials.

Based on the model proposed by Cevera and Manzoli (1996), it will be considered a specific scalar damage variable for tensile stresses, in order to obtain elastic response in compression.

The objective of this work is to evaluate the applicability of this model, based on the theory of damage, to represent the interactions of implant / crown / screw contact in dental prostheses, verifying its behavior and the stress generated in the set for different torque levels. The contact region is modeled by high aspect ratio solid finite elements, with nonlinear behavior governed by the damage model, as proposed by Manzoli et al. (2012).

2. METHODOLOGY

For this study we used a 3D biomechanical model with structural and mechanical properties corresponding to each material that composes such model: cortical bone, medullary bone, implant (pure Ti), crown (Co-Cr alloy) and retention screw (Ti-6Al-4V), presented in tab (1), still considering 950 MPa as the base value for the stress analyzes that correspond to the Ti-6Al-4V alloy yield strength.

Table 1: Properties of structures and materials used in the model.

Material	Modulus of Elasticity E (GPa)	Poisson's Ratio	Reference
Cortical Bone	13,7	0,3	Barbier et al. (1998)
Medullary bone	1,37	0,3	Barbier et al. (1998)
Implant (Ti pure)	117,0	0,30	Sakaguichi e Borgersen (1995)
Crown (alloy Co-Cr)	218	0,33	Craig (1989)
Retaining screw (Ti-6Al-4V)	103,4	0,35	Sertgoz e Gunever (1997)

A torque of 100 N.mm was considered for fixation of the components (crown / implant), as well as an oblique force representative of chewing of 133 N inclined at 30 ° and 60 ° with the vertical and displaced 2mm along the implant axis, as can be seen in Fig. 1 (GOMES, 2006).

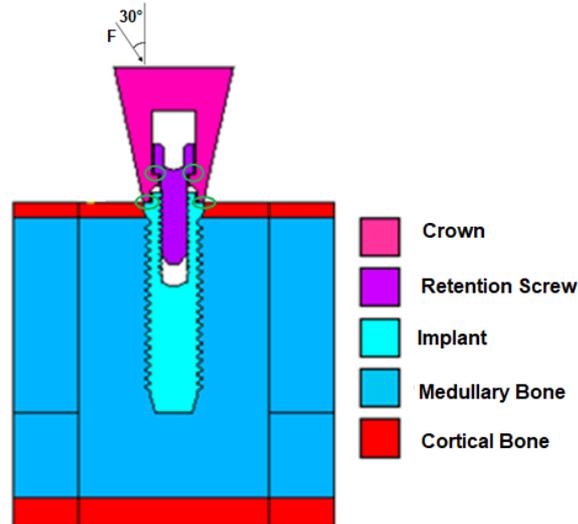


Figure 1: Biomechanical model with prominent contact areas as well as chewing force. (Gomes, 2006)

As it is a symmetrical model, only half of the 3D model was used for the simulations, according to Fig. 2 assigning orthogonal constraints on the cut face, as well as to simulate the real conditions were assigned x and y restrictions to the cortical and medullary bone and only at x for the implant, crown and screw.

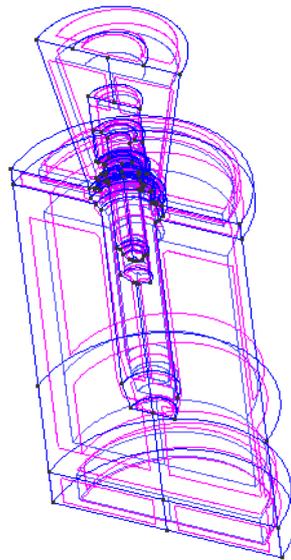


Figure 2: Symmetric 3D biomechanical model.

High aspect ratio tetrahedral solid finite elements are used as per Fig. 3 to represent the interface surface between the contacting components. A constitutive model will be employed to reproduce the behavior of these interface elements. The model is designed to represent the differential behavior in traction and compression on the contact surface allowing the separation of the components without offering virtually resistance, but at the same time prevent interpenetration movements in the event of compressive contact stresses.

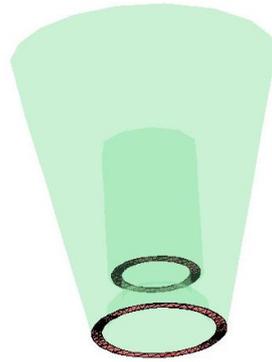


Figure 3: High aspect ratio tetrahedral solid finite elements located in the contact regions.

3. VALIDATION AND APPLICATION OF PRE-LOAD

3.1. Interface elements.

To validate the modeling, preliminary studies were performed to evaluate the constitutive model in a prismatic bar 1 mm high by 2 mm long when subjected to traction and compression.

In order for this study to be carried out, there was a need to create a contact region. Thus, this bar was divided into four equal parts. The upper parts were separated and high aspect ratio tetrahedral elements, 0.01 mm thick, were introduced between these parts.

The behavior of these interface elements is governed by the constitutive tensile damage model, with a very low tensile strength (1.10^{-05} MPa) and zero slowing parameters ($A=0$). The other mesh elements were considered with elastic-linear behavior.

First, the behavior of the interface submitted to traction was studied, in which the entire left face of the fixed bar was considered and an increasing axial displacement was imposed on the right face. Thus, it was observed the deformation of the interface elements allowing the opening, which was free of stress proving that there really was separation of the upper parts, as illustrated in Fig. 4.

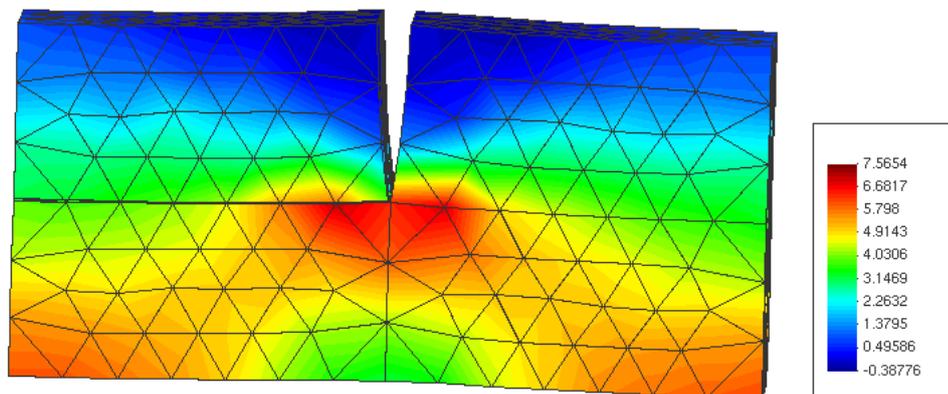


Figure 4: Normal stress due to traction.

For compression studies, an increasing approach movement was imposed on the right face, forcing axial shortening of the joint.

When required for compression, the interface model prevents interpenetration between the contact parts, generating shortening deformations Fig. 5 and uniform compression stresses throughout the assembly. In this interface model prevents interpenetration between the components in contact, generating shortening deformation and uniform compression stresses throughout the assembly.

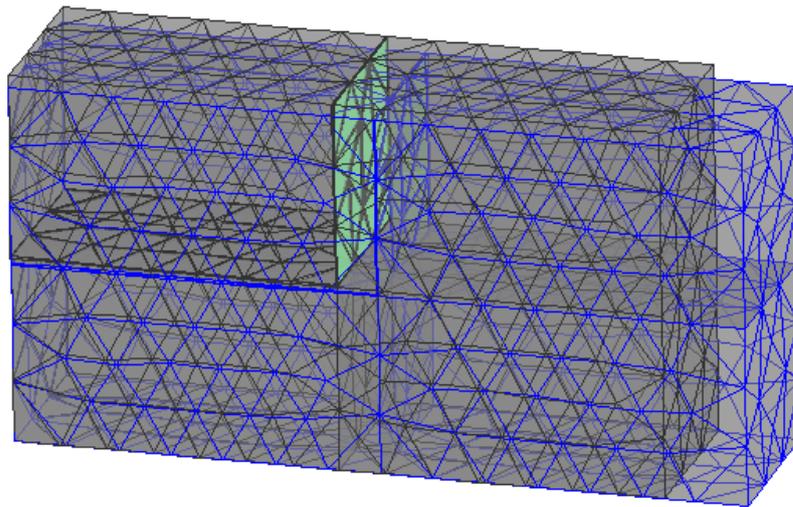


Figure 5: Deformed configuration of the model when subjected to compression.

3.2. Tightening model

To represent the effects of screw tightening, preliminary theoretical and later numerical studies were carried out to evaluate a modeling proposal that consists in the creation of a delimited region in the workpiece, subjected to an initial deformation, in order to generate the stress produced by the tightening of the screw. This region corresponds to a part of the retaining screw shaft.

To demonstrate the principle on which this method is based, consider a prismatic bar with two surfaces of linked ends, divided into three regions, as illustrated by Fig. 6.

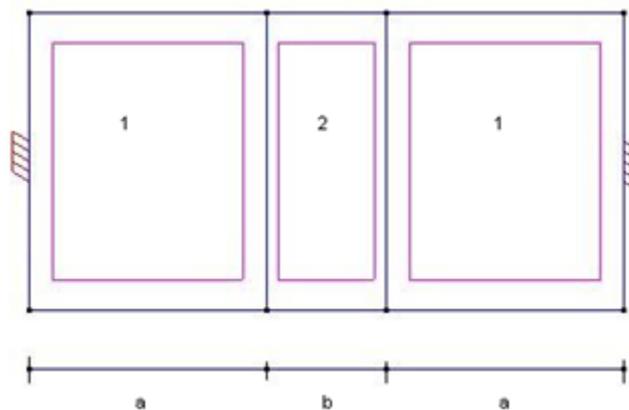


Figure 6: Schematic model for preliminary studies.

Considering the modulus of elasticity (E) of the materials and the specific deformations (ε), it is possible to express the stress (σ) from Eq. (1) and (2) respectively,

$$\sigma_1 = E_1 \cdot \varepsilon_1 \quad (1)$$

$$\sigma_2 = E_2 \cdot (\varepsilon_2 - \varepsilon_0) \quad (2)$$

From the system balance have to:

$$\sigma_1 = \sigma_2 \quad (3)$$

Substituting equation (1) in equations (2) and (3) gives:

$$\varepsilon_1 = (\varepsilon_2 - \varepsilon_0) \quad (4)$$

From the deformation compatibility condition, the length variation of the bar must be zero, therefore:

$$\Delta_L = \Delta_{L1} + \Delta_{L2} = 0 \quad (5)$$

$$\Delta_L = \varepsilon_1 \cdot (a + a) + \varepsilon_2 \cdot b = 0 \quad (6)$$

Solving the system formed by the “Eq. (4) and (6)”, obtains:

$$\varepsilon_1 = -\varepsilon_0 \cdot (1 + 2a/b)^{-1} \quad (7)$$

and

$$\varepsilon_2 = \varepsilon_1 + \varepsilon_0 \quad (8)$$

Replacing “Eq. (7)” in “Eq. (1)” gives:

$$\sigma_1 = -E \cdot \varepsilon_0 \cdot (1 + 2a/b)^{-1} \quad (9)$$

For example, assuming $E = 1.10^4$ MPa, $\varepsilon_0 = -3.10^{-2}$ mm and $a = b$, according to “Eq. (9)”, we have that the stress generated in the system is given by $\sigma_1 = 100$ MPa.

This analytical response is then contrasted with the numerical model response of the same problem by using finite elements with initial deformation.

For the numerical model it was considered the same prismatic bar (Fig. 6), which the central region imposes an initial deformation in the axial direction of the bar, in order to produce a reduction in length. Since the variation in the length of the bar is impeded by the links, the lateral regions must be elongated to compensate and shortening of the central region.

Shortening of the region under initial deformation leads to the emergence of a uniform axial tensile stress throughout the workpiece that would correspond to the effect of tightening the bolt Fig. 7. Note that the numerical answer corresponds to the analytical answer..

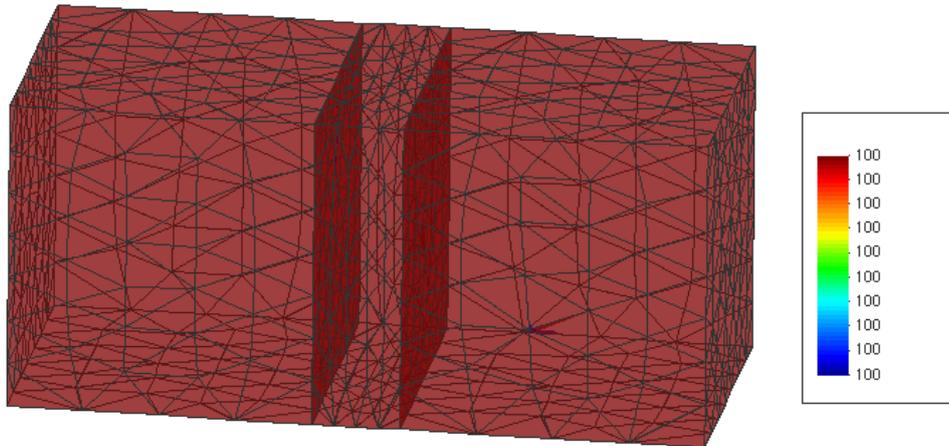


Figure 7: Uniform normal tensile stresses associated with tightening.

Thus, to represent the effects of the screw tightening, a delimited region was created in a part of the screw shaft according to Fig. 8, which will undergo an initial deformation, in order to generate the stresses produced by the tightening of that screw. Thus, the shortening of the region under initial deformation leads to the emergence of an axial tensile stress, which would correspond to the effect of screw tightening.

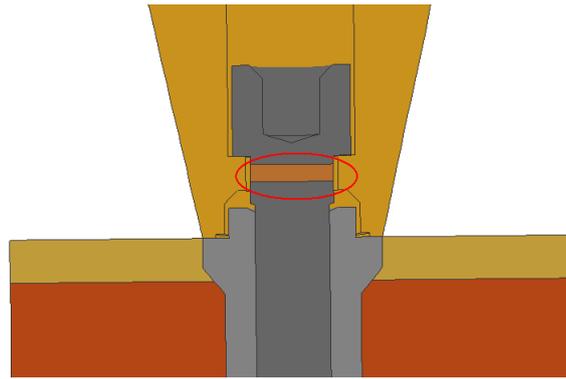


Figure 8: Detail of the boundary region created for tightening.

3.3. Preload application in the finite element model.

For a torque of 100 N.mm, the initial strain that should be imposed on the part of the screw shaft should be that which provides normal screw stresses of 129.57 MPa. Since the geometry of the problem is quite complex, this value is found through preliminary estimation and subsequent adjustment, considering the linearity of the problem.

In a preliminary analysis, estimating an imposed initial strain of $-1,43 \cdot 10^{-2}$ mm / mm, a normal stress of 105.68 MPa is obtained in the central region of the screw shaft. Thus, by linear approximation, the strain required for the stress should be $1,73 \cdot 10^{-2}$ mm / mm.

Applying the initial deformation that must be equivalent to the torque of 100 N.mm in the delimited region created by the screw, it is observed that it presented a shortening in relation to its initial state, as well as the maximum and minimum stresses (vertical) around 328.1 (maximum) and -271.4 (minimum) MPa.

4. ANALYSIS AND RESULTS

After preliminary studies of the clamping model and the constitutive model, as well as initial analyzes of normal stresses and Von Mises generated by the imposed deformation, studies of the system behavior for different load inclinations (30° and 60°) are performed. chewing, evaluating the effects of different retention screw tightening levels on the strength and functioning of the prosthesis, according to tab (2).

Table 2: Analysis at 30° and 60° .

Inclination 30°				Inclination 60°			
Torque (N.mm)	Tightening strain (MPa)	Maximum strain Von Mises throughout the set (MPa)	Opening (mm)	Torque (N.mm)	Tightening strain (MPa)	Maximum strain Von Mises throughout the set (MPa)	Opening (mm)
733	950	3064,3	-2,86E-04	733	950	3077,9	-1,73E-04
300	388,73	1254,1	-8,07E-05	300	388,73	1268,6	7,52E-05
200	259,15	836,22	-3,32E-05	200	259,15	852,41	3,51E-04
100	129,57	419,12	3,81E-05	100	129,57	574,74	2,57E-03
50	64,785	271,75	2,82E-04	50	64,785	534,03	7,27E-03
0	0	220,26	3,59E-03	0	0	557,77	1,31E-02

The displacements between crown and implant were obtained in the analyzes to study how the inclinations and grips affect these displacements.

However, despite using the recommended torque, there was a minimum opening independent of the inclination. Therefore, it would be convenient to find the ideal torque needed to avoid or minimize such opening, without producing stresses above the limit of material flow. In the present work, the reference value for the yield stress is 950 MPa (average between 800 - 1100 MPa), corresponding to the yield limit of Ti-61-4V alloy, this material of the retaining screw.

It should be noted that the negative opening value indicates a small interpenetration which, in this case, is considered with a situation without opening, that is, of elements in contact.

Based on the results achieved, it is concluded that for the 30 ° and 60 ° inclination, the recommended torque for these conditions would be 150 N.mm, however with an inevitable increase in aperture compared to the 30 ° inclination, since it is not possible to use torque higher than this value without the generated stresses exceeding the yield limit established for the material.

5. CONCLUSION

It was possible to verify that the proposed model is able to describe the effect of chewing load inclination on the system behavior. For the more inclined load (60 °), both stress and opening (crown / implant) were higher.

The model also showed that the opening (crown / implant) is reduced or even eliminated by increasing the torque value. However, in contrast, stress increase significantly and may exceed the strength limit of the implant materials.

Thus, the proposed modeling seems to be an appropriate tool to predict the ideal torque value, which would prevent (or minimize) the opening between crown and implant during chewing and at the same time produce stresses below the strength limits of the materials.

Therefore, it is concluded that the presented constitutive model, together with high aspect ratio solid finite elements located in the interface region between the different components of the prosthesis, constitutes a methodology capable of describing the contact effects on these interfaces. The model allows to represent the separation of the components, virtually without resistance, but, at the same time, prevents interpenetration movements in the case of compressive contact demands.

To represent the effects of screw tightening for the fixation of the components, the proposed technique, based on the imposition of initial deformations in a region of the shaft, proved to be quite adequate.

Thus, it was possible to represent the main aspects of interface behavior without the need for contact algorithms.

The methodology presented proved to be applicable for the three-dimensional analysis of the mechanical behavior of a dental prosthesis under different loading situations.

6. ACKNOWLEDGMENTS

To CAPES for funding the research project that gave rise to this work.

7. REFERENCES

- Cervenka, J. e Papanikolaou, V.K., 2008, "Three Dimensional Combined Fractureplastic Material Model For Concrete", *International Journal of Plasticity*, Vol. 24, pp. 2192-2220.
- Cervenra, M., Oliver, J. e Manzoli, O., 1996, "Shear Band Localization via local j2 continuum damage mechanics", *Computer Methods in Applied Mechanics and Engineering*, Vol. 25, pp. 987-1010.
- Gomes, E. A., 2006, "Efeito da ausência da passividade no sistema coroa-implante-parafuso de retenção por meio do MEF – 2D", *Dissertação (Mestrado em Odontologia) - Universidade Estadual Paulista, Faculdade de Odontologia, Araçatuba, Brasil*, 93 p.
- Grouch, F., 2010, "Biomecânica da Prótese Implanto-Suportada: Uma revisão de conceitos", *Trabalho de conclusão de curso em odontologia - Universidade Federal do Rio Grande do Sul, Porto Alegre, Brasil*, 54 p.
- Manzoli, O. L., Gamino, A. L., Rodrigues, E. A. e Claro, G. K.S., 2012, "Moideling of Interfaces in two-dimensional problems using solid finite with high aspect ratio", *Computer and Strutures*, Vol. 94, pp. 70-82.
- Patterson, E.A. e Johns, R.B., 1992, "Theoretical Analysis of the Fatigue life of Fixture Screws in Osseointegrated Dental Implants", *The International Journal of Oral & Maxillofacial Implantes*, Vol. 7, pp. 26-33.

8. RESPONSIBILITY FOR INFORMATION

The author(s) is (are) the only responsible for the printed material included in this paper.