STRESS DISTRIBUTION ON EXTERNAL HEXAGON IMPLANT SYSTEM USING 3D FINITE ELEMENT ANALYSIS

Regênio M. H. Segundo[1], Hugo M. S. Oshima[1], Isaac N. L. Silva[2], Luis H. B. Júnior[1], Eduardo G. Mota[1], Luiz F.B. Coelho[1]

[1] Department of Dental Materials, Faculty of Dentistry, Catholic Pontifical University of Rio Grande do Sul, Brazil.
[2] Faculty of Engineering, Catholic Pontifical University of Rio Grande do Sul, Brazil.

ABSTRACT
The aim of this study was to compare and evaluate strain distribution on dental implant, abutment, screw and crown virtual models in the posterior region. The analysis was performed by means of a 3D virtual model developed by the PRO-ENGINEER System (PRO-ENGINEER, PTC, Needham, MA, USA) with an external butt joint (3i Implant Innovations, Palm Beach, Florida), square headed Gold Tite® abutment retainer screw (3i Implant Innovations, Palm Beach, Florida), STA® abutment (3i Implant Innovations, Palm Beach, Florida), metal infrastructure of Ag-Pd alloy and feldspatic ceramic. The standard load was 382N at 15° angle to the implant axis, applied at 6mm from the implant center, at different observation points on the implant-screw set. The data showed that on the implant virtual model, the highest strain concentration was found at the interface between the implant platform and the abutment, and in the middle point of the 1st screw thread internal diameter, on the load application side

Key words: *finite element analysis, prosthetic joints, nonaxial loading, dental implants, biomechanic*

DISTRIBUIÇÃO DE TENSÕES EM UM SISTEMA DE IMPLANTES COM HEXÁGONO EXTERNO UTILIZANDO ANÁLISE POR ELEMENTOS FINITOS 3D

RESUMO
O objetivo deste estudo foi comparar e avaliar a distribuição de tensões em modelos virtuais de implante dental, pilar, parafuso e coroa em região posterior. As análises foram feitas através de modelo virtual 3D desenvolvido pelo sistema PRO-ENGINEER (PRO-ENGINEER, PTC, Needham, MA, USA) com sistema de conexão tipo hexágono externo(3i Implant Innovations, Palm Beach, Florida), parafuso retentor de pilar Gold Tite com cabeça quadrada(3i Implant Innovations, Palm Beach, Florida), pilar STA(3i Implant Innovations, Palm Beach, Florida), infraestrutura metálica à base de liga Ag-Pd e cerâmica feldspática. A carga padrão foi de 382N, aplicada com ângulo de 15° com o longo eixo do implante, em diferentes pontos de observação no complexo implante-parafuso. A análise dos dados mostraram que no modelo virtual do implante, o ponto de maior concentração foi encontrado na interface entre a plataforma do implante e o pilar e na metade do diâmetro interno do 1º filete do parafuso do lado da aplicação da carga.

Palavras chave: *análise por elementos finitos, conexões protéticas, cargas não axiais, implantes dentários, biomecânica.*

INTRODUCTION

Initially, titanium implants were used to rehabilitate edentulous patients, with the aim of reestablishing masticatory function. With the development of techniques and materials used in implant dentistry, other indications were added, and now implants are used for partial and single edentulism (1, 2) combining function and esthetics. At present, the lines of research are related to the type of prosthetic connection of implants, the type of retaining screw materials and the distribution of the stresses in the face of masticatory forces, which may originate prosthetic instability, single and multiple implant retaining screw fractures and even implant fractures (3). Within the context of clinical studies, an alternative for assessing failures in the implant-abutment-screw set is to use the finite element method for predicting the points with greatest potential for failure after load applications (4, 5).

The aim of this study was to make a qualitative comparison, employing finite elements and a three dimensional model, of the distribution of stresses generated by simulated loads on models of implants, abutments and metal-ceramic prosthetic crowns in the molar region.

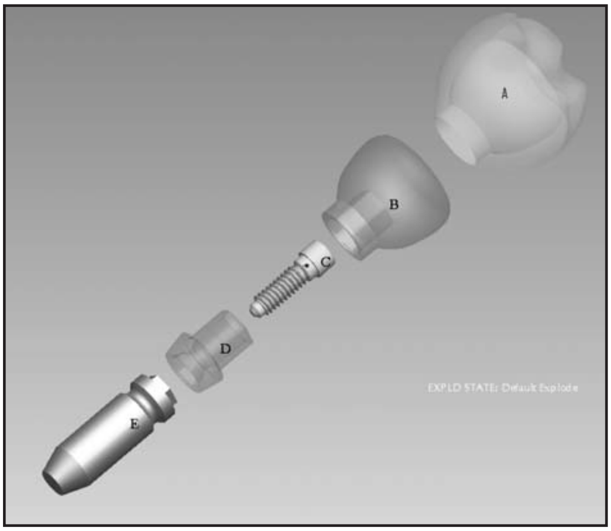

Fig. 1: expanded three dimensional image of the virtual model, A- ceramic lining, B-coping Ag-Pd, C- abutment retainer screw, D- STA abutment, E- Standard External Hexagon Implant.

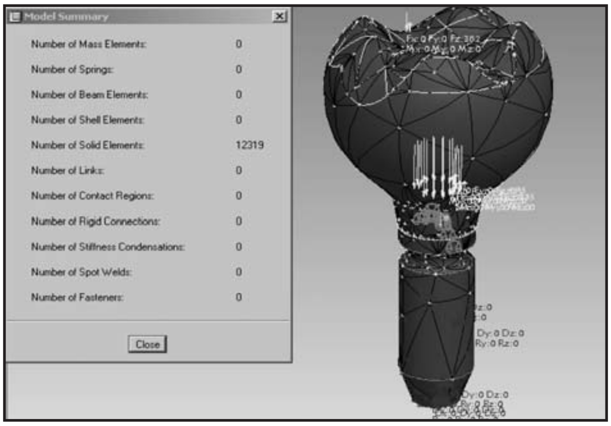

Fig. 2: Graphic representation of the virtual mesh.

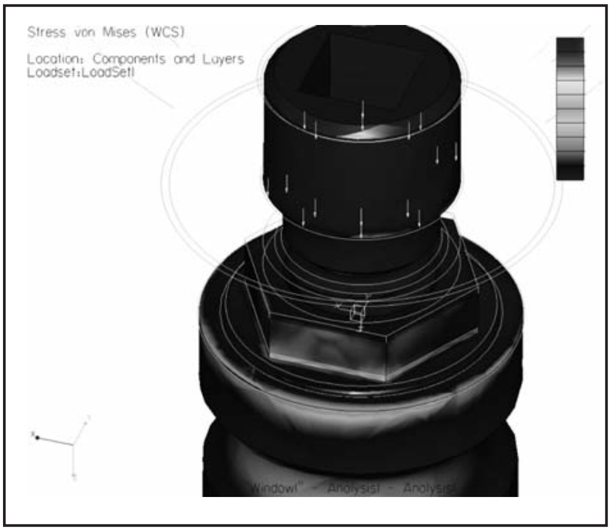

Fig. 3: Image of the stresses concentrated on the implant platform and external hexagon gaps.

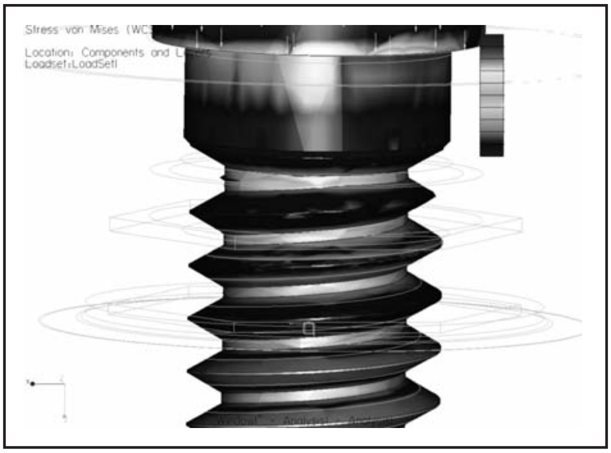

Fig. 4: Image of stresses concentrated on the midpoint of the internal diameter of the first and second screw threads of the abutment screw.

MATERIALS AND METHODS

A virtual model was constructed, with a 35N.cm^2 preload applied at the contact surface of the screw head with its respective abutment, composed of: Standard Implant 4.1mm, 4.0x11.5mm (3i Implant Innovations, Palm Beach, Florida), 2mm STA abutment (3i Implant Innovations, Palm Beach, Florida), Gold Tite square headed abutment retainer screw (3i Implant Innovations, Palm Beach, Florida) and metal infrastructure of Ag-Pd alloy (Fig 1). This model was developed by the PRO-ENGINEER (PRO-ENGINEER, PTC, Needham, MA, USA) system. It was divided into small elements, totaling 12319 tetrahedron elements, producing a Mesh (Fig 2). Load applications were simulated on the model by the PRO – MECHANICA (PTC- Needham, MA, USA) system at determined points. The load applied was 382N with a 15° angulation along the implant axis at 6mm from its center. The materials used were considered homogenous, linear and isotropic. Young's modulus of 100, 110, 100, 68.9, 95 GPa and Poisson's coefficient of 0.34, 0.34, 0.30, 0.28 and 0.33 were used for Ti_(ASTM-F67), Ti6Al4V_(ASTM-F-136), Gold alloy type 3, Feldspathic ceramic and palladium silver coping, respectively (5).

RESULTS

The analysis of the generated images, revealed a greater stress concentration at the external hexagon gaps, at the extremity of the implant platform contact with the internal abutment surface and at the midpoint of the internal diameter of the 1^{st} screw thread on the load application side (Fig 3, Fig 4).

DISCUSSION

A qualitative analysis was performed, based on a progressive visual color scale, pre-defined by the software used, ranging from dark blue to red. The results obtained in this study confirmed the biomechanical view that the abutment retainer screw is the most fragile point of the implant-prosthesis complex (6), due to the high concentration of stresses on the first pitches of the thread, which would probably lead to a future failure induced by fatigue or overload (7). Conversely the stresses induced on the external hexagon gaps and at the abutment-implant intersection are possibly of a compressive origin, and could cause the sharp angles of gaps to be burnished, and rotate the prosthetic crowns. In an attempt to minimize these problems, the development of new screw designs would be contributory (8). The new designs should include a smaller number of sharp angles at the neck and additional internal washer (9) for better stress dissipation, and implants with thicker walls to improve resistance to plastic stresses.

CONCLUSIONS

Future studies related to the finite element methods should guarantee maximum accuracy in model construction, consider anisotropic, not linear and heterogeneous materials, within the limits imposed by the software used for simulations, bearing in mind that this method of analysis is a simplified approximation of clinical situations, since it is impossible to simulate oral conditions faithfully in the virtual environment.

CORRESPONDENCE

Regênio Mahfuz Herbstrith Segundo DDS, MS, Guadalupe 90/ 601, Jardim Lindóia, Porto Alegre, RS, Brazil,
ZIP 91050 - 250 .
Fone/fax: 51- 33403304
E-mail: regesegundo@yahoo.com.br

REFERENCES

1. Carvalho W, Casado PL, Caúla AL, Barboza EP. Implants for Single First Molar Replacement: Important Treatment Concerns. Implant Dent 2006; 4:328-335.
2. Jemt T, Laney WR, Harris D, Henry PJ, Krogh PHJ, Polizzi G, Zarb GA; Herrmann I. Osseointegrated Implants for Single Tooth Replacement: A 1-year Report from a Multicenter Prospective study. Int J Oral Maxillofac Implants 1991; 1:29-36.
3. Graiton DG, Aquilino SA, Standford CM. Micromotion and dynamic fatigue properties of dental implant – abutment interface. J Prosthet Dent 2001; 85:47-52.
4. Akour SN, Fayyad MA, Nayfeh JF. Finite element analyses of two antirotational designs of implant fixtures. Implant Dent 2005; 14:77-80.
5. Geng JP, Tan KBC, Liu GR. Application of finite element analysis in implant dentistry: A review of literature. J Prosthet Dent 2001; 85:585-607.
6. Binon P. Implants and Components: Entering the New Millennium. Int J Oral Maxillofac Implants 2000; 15:77-93.
7. Binon P, Sutter F, Beaty K, Brunski J, Gulbransem H, Weiner R. et al. The Role of Screw in Implant Systems. Int J Oral Maxillofac Implants 1994; 9, Supplement: 48-63.
8. Martin WC, Woody RD, Miller BH, Miller AW. Implant abutment screw rotations and preloads for four different screw material and surfaces. J Prosthet Dent 2001; 6:24-32.
9. Versluis A, Korioth TWP, Cardoso AC. Numerical Analysis of a Dental Implant System Preloaded with a Washer. Int J Oral Maxillofac Implants 1999; 14:337-341.