INFLUENCE OF THE GEOMETRIC SHAPE OF PROSTHESIS ON THE STRESS DISTRIBUTION IN THE DENTAL IMPLANTS

Roberto Brunow Lehmann
Universidade Severino Sombra
Av. Expedicionário Osvaldo de Almeida Ramos, 280 – Centro – Vassouras – RJ.
e-mail: rbrunow@pop.com.br

Carlos Nelson Elias
Instituto Militar de Engenharia
Av. General Tiburcio, 80 – Urca – RJ.
e-mail:cnelias@bol.com.br

Jayme Pereira Gouvêa
Universidade Federal Fluminense
Volta Redonda – RJ.
e-mail:jpg@metal.eeimvr.uff.br

Abstract. *The use of computer to predict fails in dental implants have been common. The finite element analysis has been reported as an excellent method to simulate biomechanical problems with complex geometries. The purpose of this study was to compare the effects of different geometric shape of prostheses on the stress distribution in dental implants. A three-dimensional model was constructed based on commercial conical implant of 4,3mm diameter and 13mm length. The results confirm the clinical experience that the premolar region is the more critical situation of the simulated cases,, the stress in the prostheses is increasing in anterior to posterior region and the major stress in prosthetic components and the bones evaluated is to second premolar.*

Keywords*: Dental Implant, Finite Element Analysis, Stress Distribution*

1. Introduction

Currently, the finite element analysis (FEA) has been used to predict the effects of stress on implant systems, in the maxilla and mandible bone. Two-dimensional and, currently, three-dimensional models have been elaborated to analyze several implant systems and bone behavior. However, the models used were simplified in the geometry in order to minimize the number of elements and comply with hardware limitations. With the advance in technology, the models have been improved.

The prostheses are considered as a good alternative in the replacement of natural teeth. However, their manufacturing and application are very complex. The geometry of the prosthesis (Akpinar et al., 1996), dimensions (Akpinar et al., 1996; Korioth et al., 1997; Geng et al., 2001; Iplikçioglu et al.. 2002), number (Iplikçioglu et al., 2002) and disposition of implants, materials and material properties (Akpinar et al., 1996; Korioth et al., 1997; Van Oosterwyck et al., 1998; Geng et al., 2001; Meyer et al., 2001), besides the identification and analysis of the resulting efforts in the mastication, are some of the factors that can influence the results and, consequently, in the behavior (Meyer et al., 2001).

The analysis of the efforts transmitted in the interface bone-implants is essential to the determination of the implants success. The overload can cause the bony reabsorption or the failure of the implants (Stanford et al., 1999; Pilliar et al., 1991). On the other hand, under load can cause atrophy and subsequent bony loss (Stanford et al., 1999; Pilliar et al., 1991). The stresses of the bone are located in the marginal area to the implants, considered a critical area.

Many studies have been developed to better understand the stress in a process of dental occlusion. These studies, we have experimental cases 'in vitro', measurements with strain-gauge and photo-elasticity and analyses using the finite elements method. Comparative studies have been revealing contradictions among data obtained in rehearsals using photo-elasticity and strain-gauge method (Vaillancourt, Pilliar and McCammond, 1996). The literature aims a larger agreement in the comparison of results among the analyses using strain-gauge and mathematical models using the finite elements technique. Rubo and Souza (2001) report that none of the techniques have total preponderance on other. It complements to offer larger precision and reliability to the results.

The purpose of this study was to compare the effects of different geometric shape on stress distribution in the dental implants. In the simulations a conic implant with internal hexagon was used.

2. Materials and Methods

Implant (Fig.1) and components were modeled on a personal computer (Pentium 4 Processor with 1.6 GHz; 1GB of DDR266MHz RAM Memory and hard disk with 40GB) using a finite element program (ANSYS version 5.7). In an attempt to simulate a simplified mandible segment, a cancellous core surrounded by a 1,3mm thick cortical layer was modeled around the implants. The overall dimensions of this block were 23,4mm in height, 25,6mm in mesiodistal length, and 9mm in buccolingual width. The implant was modeled with 13,4mm in height and 5,4mm of diameter.

Small simplifications in the geometry were adopted objectifying to reduce the elements number of the model. Each component was modeled separately to allow the individual visualization of the components to verify the stress levels with base in the different colors scales supplied by the program.

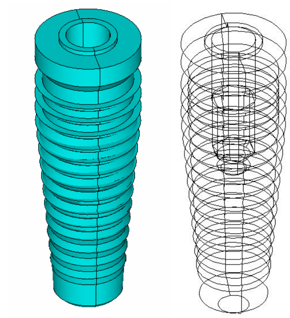

Figure 1 – Conical implant

2.1. Material Properties

All materials used in the models were considered homogeneous, isotropic and linearly elastic. Another consideration was admit an ideal osseointegration of the implants. In fact, the osseointegration happens in 80% of the implants surface.

To define the material properties of neoformed bone, it was adopted an intermediate value between the cortical and cancellous bone. The implant used is made in titanium ASTM 4. The material properties of the cortical and cancellous bone were defined according to the literature (Lehmann, Elias e Gouvêa, 2002; Lehmann, 2005). Table 1 presents the material properties used in the models.

Table 1. Mechanical properties of the used materials.

MATERIAL	YOUNG´S MODULUS GPa	POISSON´S RATIO
Titanium	110.00	0.35
Cortical bone	15	0.3
Cancellous bone	1.5	0.3
Neoformed bone	8.25	0.3

2.2. Modeling

It was used the three-dimensional element SOLID 92 (Ansys, 2002) in the models. The element is defined by ten nodes having three degrees of freedom at each node: translation in the directions x, y and z. These directions in the system of nodes coordinates correspond to the radial, axial and tangential directions, respectively. Another advantage of the element SOLID 92 is to tolerate irregular forms without loss precision.

Each generated model presented approximately 50,000 elements, within ten independent volumes: cortical bone, cancellous bone, neoformed bone with cortical interface, neoformed bone with cancellous interface, implant, abutment, screw of abutment fixation, copping, screw of copping fixation and the prosthesis (teeth).

2.3. Load and Constraints

The load applied was 100N to teeth with oclusal table and 50N without oclusal table. The load was applied in the superior surface of the teeth to obtain a centric oclusal. The movement restrictions were applied in the areas distal-end, in all the directions.

Starting from all these definitions, it was possible to use the program to calculate the von Mises stress in the bones and in the components of the implant systems used.

3. Results

The values of stress obtained are shown in the Tab. 2. The stresses transmitted for the bones were analyzed, as well as, the stress in implant and its components (abutment, copping and fixation screws).

Table 2. Maximum von Mises Stresses (MPa)

Components \ Model	31	32	33	34	35	36	37
Prostheses	**3**	**11.6**	**7.7**	**15.9**	**16.9**	**31.5**	**35.2**
Screw of copping fixation	**1.9**	**26.5**	**19.2**	**26**	**53.7**	**32.4**	**31.6**
Copping	**18.2**	**10.8**	**25.7**	**37.3**	**44.9**	**35.6**	**44.9**
Abutment	**25.3**	**35.3**	**37.4**	**51.9**	**60.2**	**30.9**	**51.6**
Screw of abutment fixation	**14**	**33.4**	**18.7**	**29.3**	**34.2**	**21.4**	**29.1**
Implant	**37.6**	**51.4**	**50**	**69.7**	**85.2**	**38.3**	**66**
Neoformed bone/Cancellous	**4.1**	**5.5**	**5.6**	**8.5**	**9.8**	**4.7**	**7.5**
Neoformed bone/Cortical	**22.7**	**29**	**28.5**	**35**	**43.1**	**15.3**	**26.7**
Cortical bone	**6.7**	**9.1**	**9**	**12.4**	**13.8**	**5.8**	**11.5**
Cancellous bone	**1.1**	**1.3**	**1.3**	**2.1**	**2.6**	**1.1**	**1.7**

The results found for all the simulations indicated that the point of cortical bone with larger stress concentration is the marginal area of the implants (neck). The stress values varied according to the prosthesis geometry.

The stress in the cortical bone was smaller in the models for central incisors (figure 2). The greatest stress in cortical bone was found in one of the second pre-molar models (figure 3). The differences found were significant.

The values found of the stresses in the cancellous bone were smaller that 3MPa, for all the models analyzed.

When analyzing the bone layer formed on surface implant (neoformed bone), was possible to observe that the stress were bigger on neoformed bone with cortical interface in the second pre-molar. The results indicate that the fail is possible in this region.

Implant, abutment and screws presented higher stress levels for pre-molar models when compared to the central incisor models.

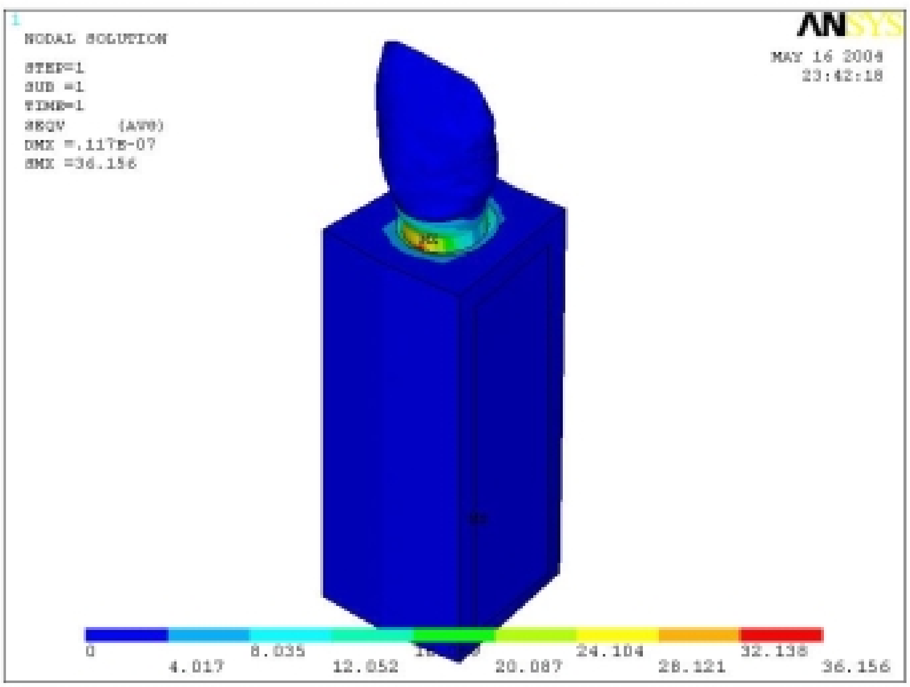

Figure 2 – von Mises stress (MPa) distribution on the central incisor (model 31)

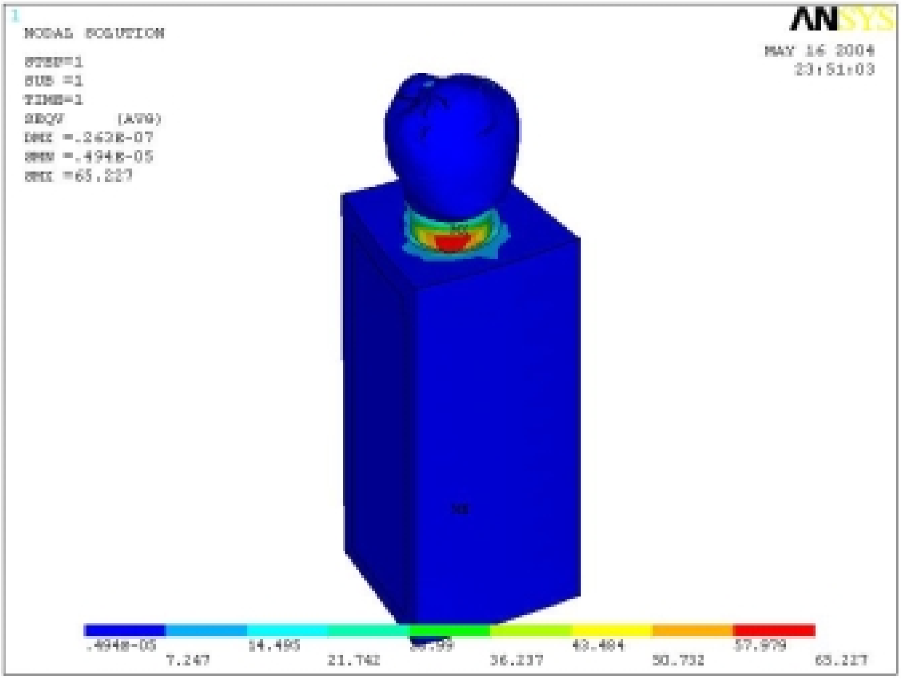

Figure 3 – von Mises stress (MPa) distribution on the second premolar (model 35)

4. Discussion

Clinical studies reported significant bone loss around the implant neck of failing implants, and various hypotheses have been proposed to explain this bone reaction (Iplikçioglu et al., 2002; Keyak et al., 1993). Inappropriate loading causes excessive stress in the bone around the implant and may result in bone resorption. Therefore, it is valuable to investigate the stresses in the bone and their relation to different geometric shapes of the prosthesis in simulates cases by finite element analysis. In these works, it is considered the existence of threads on the implant and neoformed bone on the surface of the implant.

Stegaroiu et al. (1998) concluded that the quality of the trabecular bone influences in the determination of the present stress in the cortical bone and the cylindrical implant with threads are more appropriate than the implant without threads. In the present work, a commercial conical implant was used. All the values used of the dimensions of the implant and prosthetic components are real and made by the Connection Company that manufactures the implants.

It can be observed that for the presented results, the geometric shape of the prosthesis, as well as the load and constraints applied, the properties of the materials and even the existence of the bone layer formed in the surface of the implants, influence in the stress transmitted to the bone. That indicates the need to develop models more and more complex, in order to reproduce to the maximum the reality of the mechanical behavior of the implant, prostheses, bone or any other element modeled.

Lehmann et al. (2005) reported that the incidence of fractured Branemark implants was low, but 90% of implant fractures occurred in the posterior region.

The critical cases are presented by the second pre-molar models, compared to the lower stress levels presented by the central incisor models.

5. Conclusion

The results obtained through the presented simulations, suggest:

1. The results confirm the clinical experience that the premolar region is the most critical for the simulated cases;
2. The stress in the prostheses increases from the anterior to the posterior regions;
3. The major stress in prosthetic components and the bones evaluated is to second premolar.

6. References

Akpinar I, Demirel F, Parnas L, Sahin S., 1996, "A comparison of stress and strain distribution characteristics of two different rigid implant designs for distal extension fixed prostheses", Quintessence Int, Vol.27, pp. 11-17.

Ansys Element Reference, version 5.7, 2002.

Brunski JB, 1999, "In Vivo Bone Response to Biomechanical Loading at the Bone/Dental-Implant Interface." 15th International Conference on Oral Biology. Vol. 13.

Geng JP, Tan KBC, Liu GR., 2001, "Application of finite element analysis in implant dentistry: A review of the literature", The Journal of Prosthetic Dentistry. Vol.85, pp. 585-598.

Iplikçioglu H, Alça K., 2002, "Comparative evaluation of the effect of diameter, length and number of implants supporting three-unit fixed partial prostheses on stress distribution in the bone", Journal of Dentistry, Vol. 30, pp. 41-46.

Keyak JH, Fourkas MG, Meagher JM, Skinner HB., 1993, "Validation of na automated method of three-dimensional finite element modelling of bone", Journal of Biomedical Engineering, Vol.15, pp. 505-509.

Korioth TW, Versluis A., 1997, "Modeling the mechanical behavior of the jaws and their related structures by finite element (FE) analysis", Crit Rev Oral Biol Med. Vol.8, pp. 90-104.

Lehmann, RB, Elias CN, Gouvêa JP., 2002, "Simulação de torção em limas endodônticas fazendo análise por elementos finitos", Congresso Nacional de Engenharia Mecânica – Brasil.

Lehmann, RB, 2005, "Avaliação tridimensional para avaliação de tensões em implantes dentários pelo método de elementos finitos", Thesis, Fluminense Federal University.

Meyer U, Vollmer D, Runte C, Bourauel C, Joos U., 2001, "Bone loading pattern around implants in average and atrophic edentulous maxillae: a finite-element analysis", Journal of Cranio-Maxillofacial Surgery, Vol. 29, pp. 100-105.

Pillar RM, Deporter DA, Watson PA, Valiquette N., 1991, "Dental implant design—effect on bone remodeling", J Biomed Mater Res, Vol.25, pp. 467-83.

Rubo JH, Souza EAC., 2001, "Métodos Computacionais Aplicados à Bioengenharia: Solução de problemas de carregamento em próteses sobre implantes", Revista da Faculdade de Odontologia de Bauru, Vol.9, pp. 97-103.

Stanford CM, Brand RA., 1999, "Toward na understanding of implant occlusion and strain adaptive bone modeling and remodeling", Journal of Prosthetic Dentistry, Vol.81, pp. 553-561.

Stegaroiu R, Nishiyama S, Miyakawa O, Kusakari H., 1998, "Influence of Prosthesis material on stress distribution in bone and implant: a 3-dimensional finite element analysis" The International Journal of Oral & Maxillofacial Implants, Vol.13, pp. 781-790.

Tada S, Stegaoiu R, Kitamura E, Miyakawa O, Kusakari H., 2003, "Influence of implant design and bone quality on stress / strain distribution in bone around implants: A 3-dimensional finite element analysis", The International Journal of Oral & Maxillofacial Implants, Vol.18, pp. 357-367.

Vaillancourt H, Pillar RM, McCammond D., 1996, "Factors affecting crestal bone loss with dental implants partially covered with a porous coating: a finite element analysis", Int J Oral Maxillofac Implants, Vol.11, pp. 351-9.

Van Oosterwyck H, Duyck J, Vander Sloten J, Van der Perre G, De Cooman M, Lievens S, 1998, "The influence of bone mechanical properties and implant fixation upon bone loading around oral implants", Clin Oral Implants Res. Vol.9, pp. 407-18.

7 . Responsibility Notice

The authors are the only responsible for the printed material included in this paper.