INFLUENCE OF CANTILEVER ON STRESS DISTRIBUITION IN BONE AROUND DENTAL IMPLANTS: A THREE-DIMENSIONAL FINITE ELEMENT ANALYSIS

Roberto Brunow Lehmann
Universidade Federal Fluminense – Av. dos Trabalhadores, 280 – Volta Redonda – RJ.
e-mail: rbrunow@pop.com.br

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

Abstract. *The use of 3D model to simulation dentistry problems with finite element method has been reported. The purpose of this study was to evaluate the stress distribution in the cortical bone under different shape of dental implants in cantilever loading. The models were constructed with commercial conical and cylindrical implants of 4.3 mm diameter and 11mm length using ANSYS program. All materials used in the models were considered to be isotropic, homogeneous and lineary elastic. The elastic properties, the loads and constraints used were taken from the literature. The highest values of stress are found when the cylindrical implant was used. Therefore, the results indicate that the use of conical implant in patients with cantilever structure is more indicated if compared with the cylindrical implant.*

***Keywords**: Dental Implants, Cantilever, Finite Element Analysis.*

1. INTRODUCTION

The use of computer to predict fails in dental implants has been common. The element finite analysis was been reported to simulate biomechanical problems with complex geometries.

The load analysis in the interface bone-implants is essential to the determination of the implants success. The overload can cause the bone reabsorption or the failure of the implants (Sahin et al., 2002; Stanford et al.,1999; Pilliar et al., 1991). On the other hand, under load can cause atrophy and subsequent bone loss (Stanford et al.,1999; Pilliar et al., 1991). The stresses in 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 have experimental in vitro measurement 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 et al., 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) conclude that none of the techniques have total preponderance on other. It complements to offer larger precision and reliability to the results.

Some authors (Lewinstein et al., 2003; Brosky et al., 2003; Parel et al., 2001; Iplikçioglu and Akça, 2002) studied the short cantilever situations and confirm to prevent its use because the constant implant screw loosening, but the implant cantilevers are often clinically necessary for occlusal support and esthetics.

Brosky et al (2003) commented the effects of cantilever length on stresses transferred to bone by the implant-supported prosthesis. Their findings revealed that cantilever length can influence forces delivered to the implants and bone and can directly affect marginal bone loss. Rangert et al (1989) addressed the fundamentals of the mechanical parameters that determine the load on implant units. They demonstrated that prosthesis design and implant placement have a significant influence on bone stress, as well as on screw attachment.

The purpose of this study was to evaluate the stress distribution in the cortical bone when different shapes of dental implants in cantilever loading were used.

2. MATERIALS AND METHODS

Implant and components were modeled on a PC (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 bone 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.

2.1. Material properties

All materials used in the models were considered to be isotropic, homogeneous and elastic. The implant used is of titanium ASTM grade 4. The material properties of the cortical and cancellous bone were defined according to the literature (Lehmann et al. 2005; Lehmann et al. 2006), “Tab. 1”.

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. Elements and Nodes

The finite element used was SOLID 92. This element allows the analysis of a three-dimensional geometry (Ansys Element Reference). 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 51,609 elements to cylindrical model “Fig.1” and 57,403 elements to conical model. All models have 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).

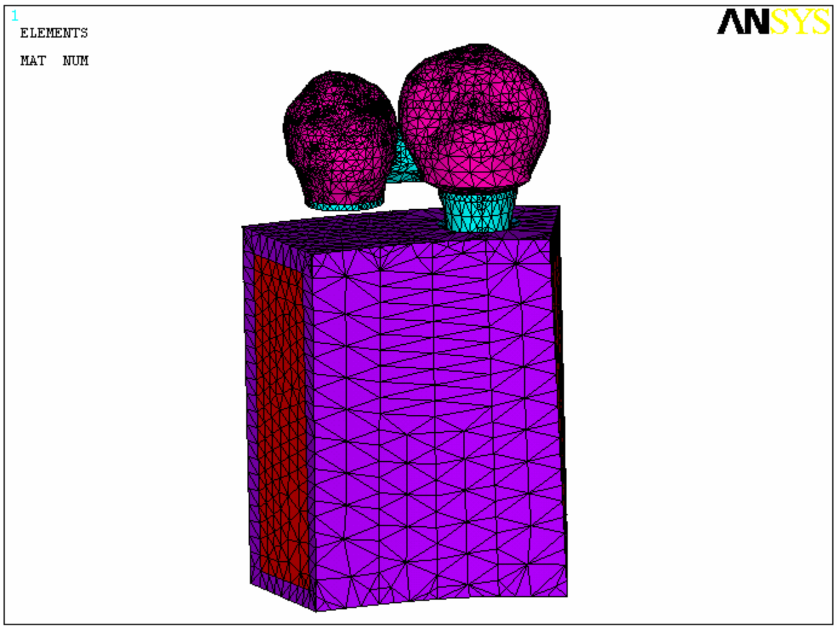


Figure 1 – Three-Dimensional Model used

2.3. Load

The load applied was 100N to each tooth. 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.

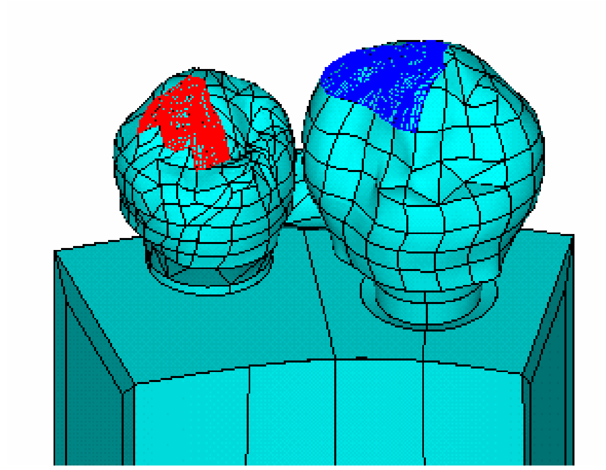
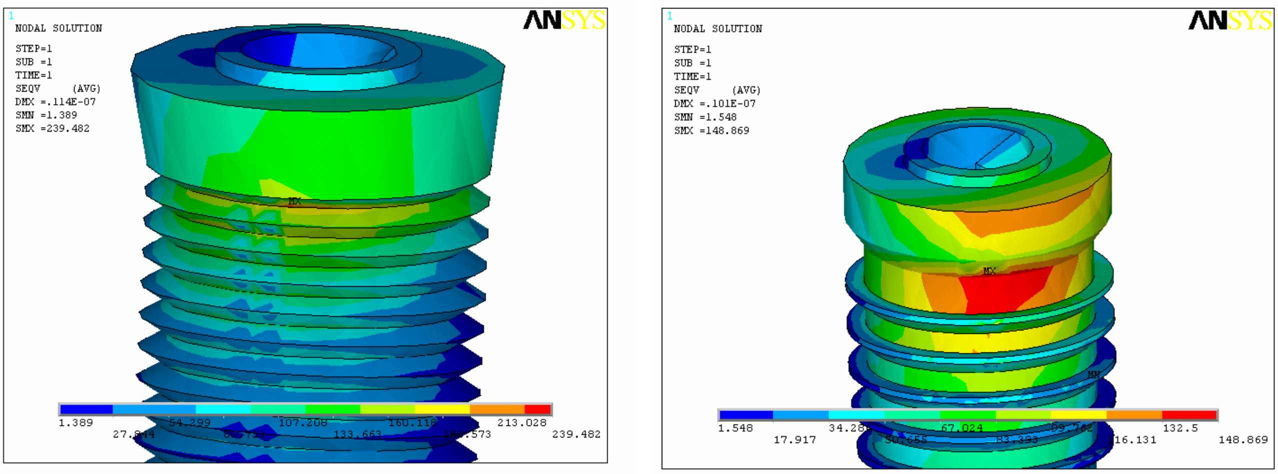


Figure 2 – Loads applied in model proposed

3. RESULTS AND DISCUSSION

For analysis in all simulations, the von Mises stress was used. The values obtained of stress are shown in the figures 3 and 4. The stresses transmitted for the bones were analyzed, as well as, the stress in implant and its components.

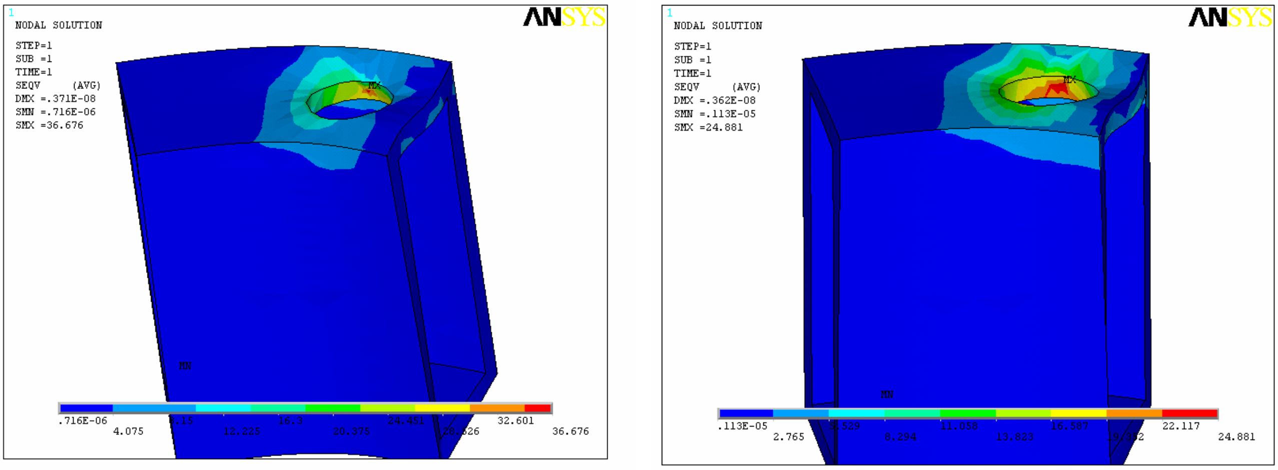
The results found for both the simulations indicated that the point of cortical bone with larger stress concentration is the marginal area of the implants (neck). To implants analyzed the major stress are found to cylindrical shape "Fig. 3" always near of the first thread. The stress values varied according to the implant used. The stress in the cortical bone with conical implant was smaller than in cylindrical implant, "Fig. 4". The differences found were statistical significant. The values found of the stresses in the cancellous bone were smaller that 7 MPa, for both the models analyzed.



(b)

(a)

Figure 3 – von Mises stress (MPa) to implant: (a) cylindrical; (b) conical.



(b)

(a)

Figure 4 – von Mises stress (MPa) to cortical bone with: (a) cylindrical implant; (b) conical implant.

Barbier et al. (1998) found 27,5 MPa to cortical bone and 2,9 MPa to cancellous bone by three-dimensional model using finite element analysis. The results found are in accordance with the author.

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 bone reabsorption. Therefore, it is valuable to investigate the stresses in bone and their relation to different geometric shapes of prosthesis in simulates cases by finite element analysis. In these works are considered the existence of threads on implant and the neoformed bone on surface implant.

The quality of the cancellous bone influences in the stress in the cortical bone and the cylindrical implant with threads is more appropriate than implant without threads (Stegaroiu et al., 2003). All the values used of the dimensions of the implant and prosthetic components are real and made by the Connection Company (São Paulo, SP) that manufactures the implants. Can be observed that for the presented results, that the geometry 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 for the bone. That indicates the need to develop models more and more complex, looking for to reproduce to the maximum the real mechanical operation of the implant, of the prostheses, of the bone or any other element in the analysis proposal.

4. CONCLUSION

The results obtained through the presented simulations, suggest:

1. The results confirm the clinical experience that the point of cortical bone with larger stress concentration is the marginal area of the implants;
2. The better situation occurred when the conical implant was used because it transmitted less stress to the cortical bone than cylindrical implant;
3. The conical implant is more indicate when compared to cylindrical implant.

4. REFERENCES

Akça, K., Iplikçioglu, H., 2002, "Finite element stress analysis of the effect of short implant usage in place of cantilever extensions in mandibular posterior edentulism.", Journal of Oral Rehabilitation, Vol.29, p.350-356.

Ansys Element Reference, version 5.7.

Barbier L., Sloten J. V., Krzesinski G., Et al., 1998, "Finite Element Analysis of Non Axial Versus Axial Loading of Oral Implants in the Manible of the Dog.", Journal of Oral Rehabilitation, Vol. 25, p. 847-858.

Brosky, M. E., Korioth, T. W. P., Dent, C., Et al., 2003, "The anterior cantilever in the implant-supported screw-retained mandibular prosthesis.", The Journal of Prosthetic Dentristry, Vol.89, No.3.

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.

Lehmann, R.B., 2005, "Avaliação tridimensional de tensões em implantes dentários pelo método de elementos finitos", Theses, Fluminense Federal University.

Lehmann, RB, Elias CN, Gouvêa JP., 2006, "Análise tridimensional de três implantes dentários cônicos interligados", Congresso Nacional de Engenharia Mecânica – Brasil.

Lewinstein, I., Fanor, Y., Pilo, R., 2003, "Abutment positioning in a cantilevered shortened dental arch: A clinical report and static analysis", The Journal of Prosthetic Dentistry, Vol. 89, No. 3.

Parel, S. M., Branemark, P. I., Ohrnell, L. O., Et al., 2001, "Remote implant anchorage for the rehabilitation of maxillary defects.", The Journal of Prosthetic Dentistry, Vol. 86, No. 4, p.377-381.

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

Rangert, B.; Jemt, T.; Lars, J., 1989, "Forces and moment on Branemark implants." International Journal Oral Maxillofacial Implants, Vol.4, p.241-247.

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.

Sahin, S., Çehreli, M. C., Yalçin, E., 2002, "The influence of functional forces on the biomechanics of implant-supported prostheses – A review", Journal of Dentistry, Vol.30, p. 271-282.

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.

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.

5. RESPONSIBILITY NOTICE

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